HEIDENHAIN

MANUALplus 620

The Contouring Control for CNC and Cycle Lathes

September 2012

Start smart

For many years now, the MANUALplus has been proving itself in daily use on cycle lathes and has earned a reputation for convenient manual machine operation. Application-oriented cycle programming enables the machinist to create and edit programs rapidly and efficiently on the lathe. The introduction of the MANUALplus 620 extends the area of application to single-spindle CNC lathes. With the smart.Turn operating mode, HEIDENHAIN has made yet another step forward toward greater ease of use. Easily understandable program entry in forms, default setting for global values, numerous selections and straightforward graphic support ensure fast and easy operation.

The new smart.Turn interface is based on the proven HEIDENHAIN-DIN PLUS. Because smart.Turn produces DIN PLUS programs. It provides both the NC programmer and the machine operator with all relevant information during program run.

This brochure describes the functions and specifications of the MANUALplus 620 with NC software 54843x-01.

Contents

The MANUALplus 620...

Where can it be used?	Compact and versatile – MANUALplus 620, the control for CNC and cycle lathes	4
What does it look like?	Well designed and user friendly – The MANUALplus 620 in dialog with the user	6
What can it do?	Universally applicable – The right programming mode for every task	8
	Easy machining with cycles – Preprogrammed machining steps – From single parts to series	10
	Well thought out, simple and flexible – Simple programming with smart.Turn – NC program at the push of a button with TURN PLUS	14
	Describing and importing contours – ICP interactive contour programming	18
	Realistic testing before machining – Graphic simulation	20
	Expandable for complex tasks – Full-surface machining including the C axis and Y axis – Turning, drilling and milling in one setup – Working in a tilted plane with the B axis	22
	Fast availability of tool data and cutting data – MANUALplus tool database and technology database	28
	Workpiece measurement – Setup, presetting and measuring with touch trigger probes	30
	Tool measurement – Measuring length, radius and wear directly in the machine	31
	Open for communication – Fast data transfer with the MANUALplus 620 – Display any file formats on the control screen – The DataPilot MP 620 programming station – Fast availability of all information	32
... At a glance	Overview – User functions – Options – Accessories – Specifications	36

Compact and versatile
– MANUALplus 620, the control for CNC and cycle lathes

Thanks to its flexible design and numerous programming features, MANUALplus 620 always gives you optimum support. Regardless of whether you are manufacturing single parts or batches, simple or complex workpieces, the control always adapts to the needs of your company. The MANUALplus 620 is characterized by its simple operation and programming. It is quickly learned and requires minimum training time.

The MANUALplus 620 was conceived both for CNC and cycle lathes. It is suitable for horizontal and vertical lathes. The MANUALplus supports lathes with simple tool holders and lathes with tool turrets. The tool carrier of horizontal lathes can be located in front of or behind the workpiece.

The MANUALplus supports lathes with spindle, one slide (X and Z axis), C axis or positionable spindle, driven tools and machines with a Y axis and B axis.

MANUALplus 620 for cycle lathes
On the MANUALplus 620, rework or simple tasks can be done in the same way as on a conventional lathe. You move the axes in the normal manner by turning the handwheels. For difficult cuts like tapers, undercuts or threads, the cycles of the MANUALplus are available.

For small and medium-size production runs, cycle programming will increase your revenue. When machining the first workpiece, you can store the machining cycles, and already save valuable time when machining the second workpiece.

For increased requirements and complex machining tasks, you will profit from the new programming mode smart.Turn.

MANUALplus 620 on a cycle lathe

MANUALplus 620 for CNC lathes
Regardless of whether you are turning simple parts or complex workpieces, the MANUALplus 620 provides you with the benefits of graphical contour input and convenient programming with smart.Turn.

Programming with variables, controlling special machine components, or using externally created programs, etc. is no problem: simply switch to DIN PLUS. This programming mode helps you solve all your special tasks.

MANUALplus 620 on a CNC lathe

Well designed and user friendly
– The MANUALplus 620 in dialog with the user

The screen
The 12.1-inch TFT color flat-panel display shows a clear overview of all relevant information for programming, operating and inspecting the machine tool and control such as program blocks, comments and error messages.

During program input the required parameters are illustrated in help graphics, and during test run the MANUALplus simulates the cutting process on the screen. During program run the screen displays information on the tool position, the rotational speed, the feed rate and the utilization of the drives as well as further information on the machine status.

The positions of the tool are shown in large characters. The respective distance-to-go, the feed rate, the spindle speed and the ID number of the current tool are also clearly visible. A moving-bar diagram shows the current utilization of the spindle and the axis drives.

The keyboard
The MANUALplus needs very few keys. Easily understood symbols clearly indicate the functions.

The keys on the numeric keypad are used both for data input and for selecting the functions. The menu window displays the available functions graphically. The function keys below the screen are used to modify the selected functions, assume position and technology values, and control the data input.

The machine operating panel
The machine operating panel is designed by the machine tool builder. Your machine panel might look like this: handwheels for saddle and cross slide, joystick for continuous axis movement, rotary switch for the handwheel transmission ratio, rotary switch for spindle CCW/CW/stop, buttons for cycle start/stop and an emergency stop switch.

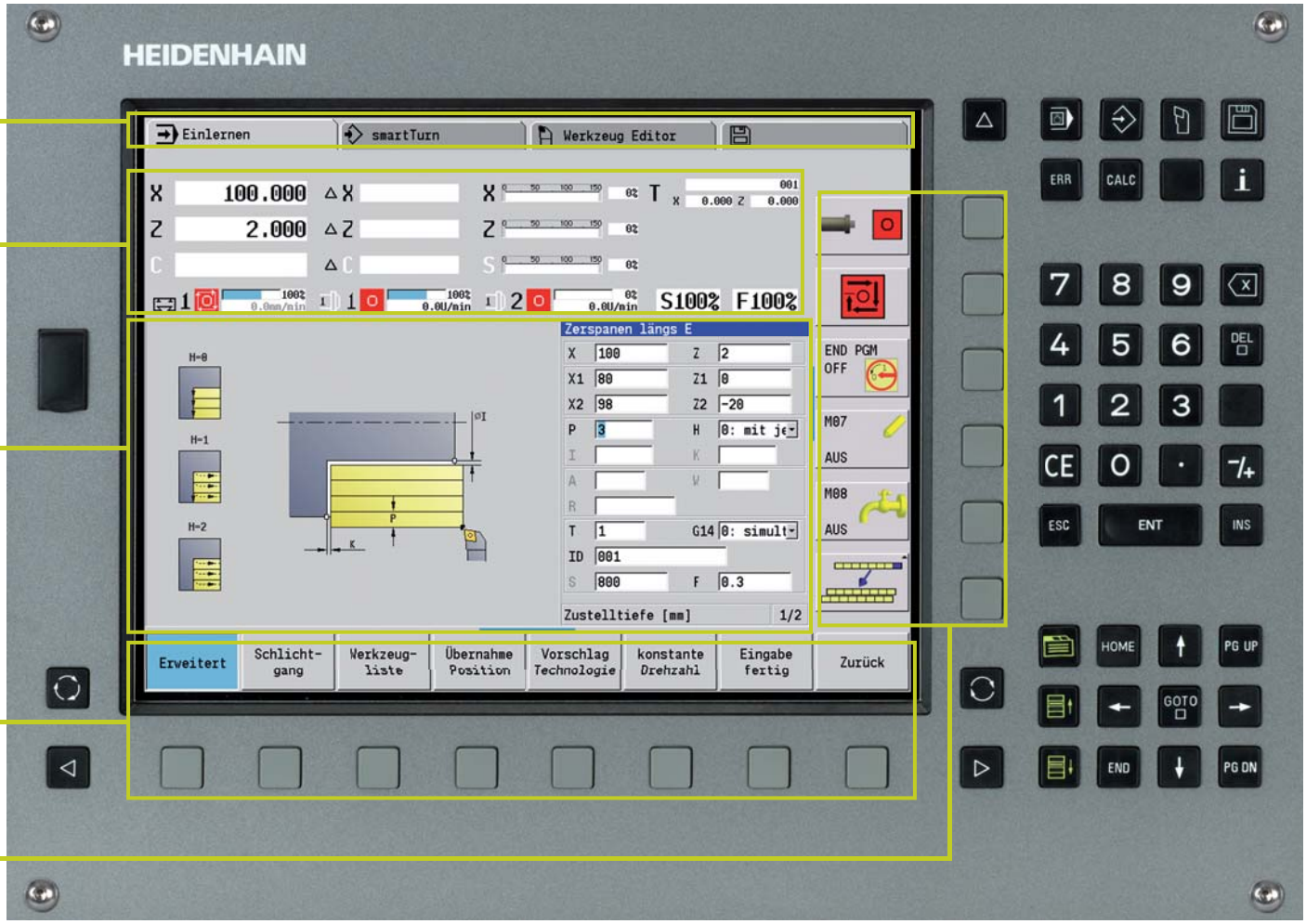
Operating modes

Display of the machine status. The display is configurable. You can choose a suitable function for each of the 16 fields, and save different display assignments for the automatic and manual mode.

Straightforward input forms for cycle programming, smart.Turn programming or DIN PLUS programming. The input parameters are illustrated in help graphics during NC programming.

Unambiguous function keys for NC programming

PLC function keys for machine components



Operating mode keys

- Machine operating modes
- Programming modes
- Tables for tool data and technology data
- Parameters, file management, transfer, diagnostics

smart keys

- Switch to the next detail input form
- Previous/next group

Navigation keys

- Screen/page up/down
- Go to beginning of program/list or to end of program/list

Special keys

- Activate special functions, such as input options or input of characters as on a cell phone
- Calculator
- Call up messages and errors
- Clear error from machine operating modes
- Switch the help graphics between outside/inside machining (cycle programming)

Universally applicable
– The right programming mode for every task

With its different programming operating modes, **cycle programming, smart.Turn** and **DIN PLUS**, the MANUALplus 620 always gives you optimum support for your tasks.

Regardless of whether you want to rework a workpiece or machine a single part on a cycle lathe, the cycles of the MANUALplus 620 will simplify your work. For manufacturing batches, you create a cycle program by saving the cycles during the machining of the first workpiece. The effective programming mode smart.Turn for fast and easy NC program creation helps you take care of more demanding tasks.

Manual
- Simple, non-repetitive machining tasks
- Rework
- Thread repair

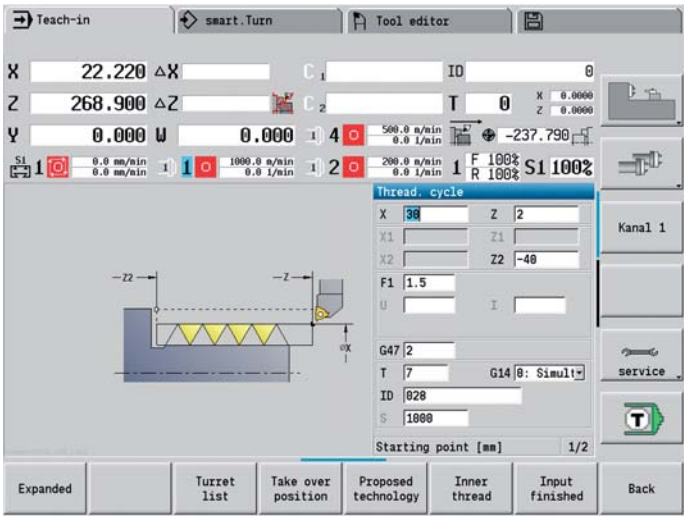
Teach-in
- Manual-oriented machining of small and medium-size production runs
- Graphic description of complex contours

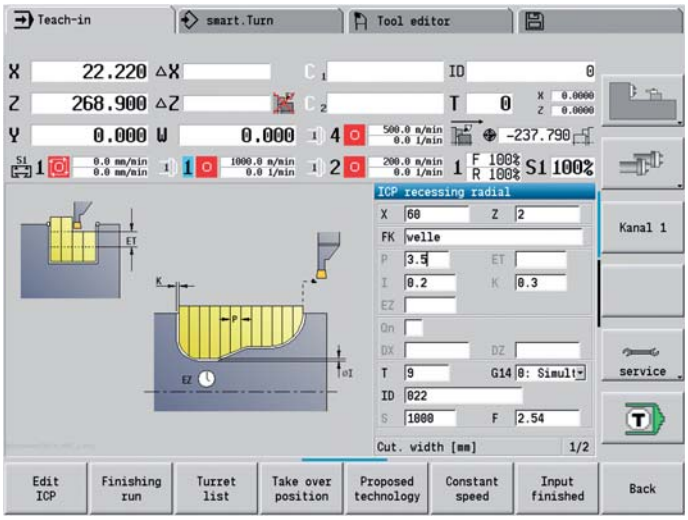
Machining with cycles

Define the cycle
↓
Simulate the cycle
↓
Machine the workpiece

Machining with cycles

Define the cycle
↓
Simulate the cycle
↓
Machine the workpiece
↓
Save the cycle
↓
Finished cycle program



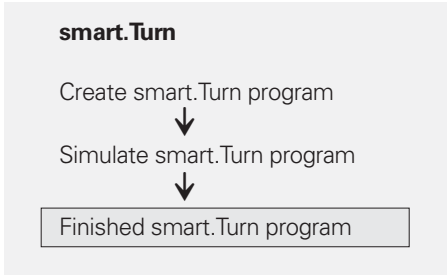


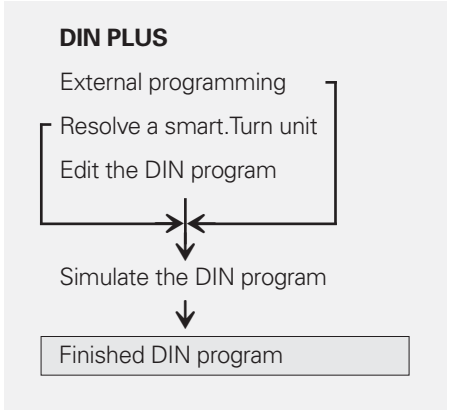
On CNC lathes, the best method to create a program is to use smart.Turn. This programming mode from HEIDENHAIN is based on input forms and allows you to create structured, easy-to-read NC programs in which you can even store all set-up information needed to machine the workpiece.

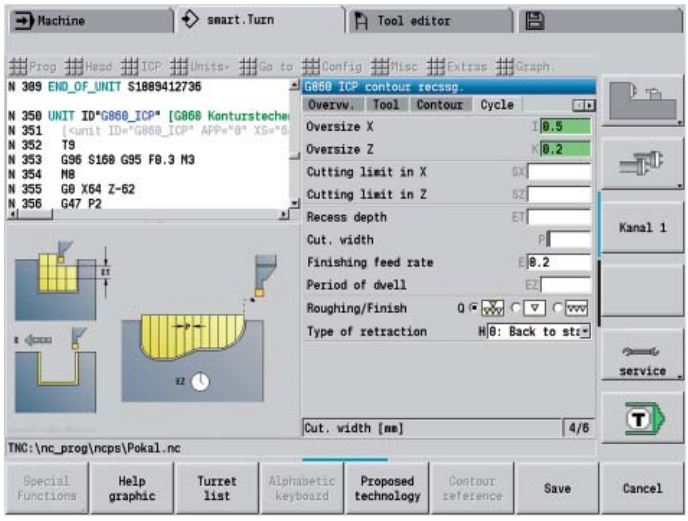
If you want to use the variable programming feature or externally created NC programs, or if you have to cope with special requirements, DIN PLUS will give you optimum support.

smart.Turn
- Convenient smart.Turn programming
- Graphic contour description
- Numerous machining units
- Compatible with cycle programs
- Clearly structured and easy-to-read programs
- All required data contained in the respective working block

DIN PLUS
- Familiar G-code programming
- Graphic contour description
- Numerous fixed cycles
- Variable programming and subprograms
- Resolving smart.Turn units into G-code commands
- Compatible with externally created DIN programs







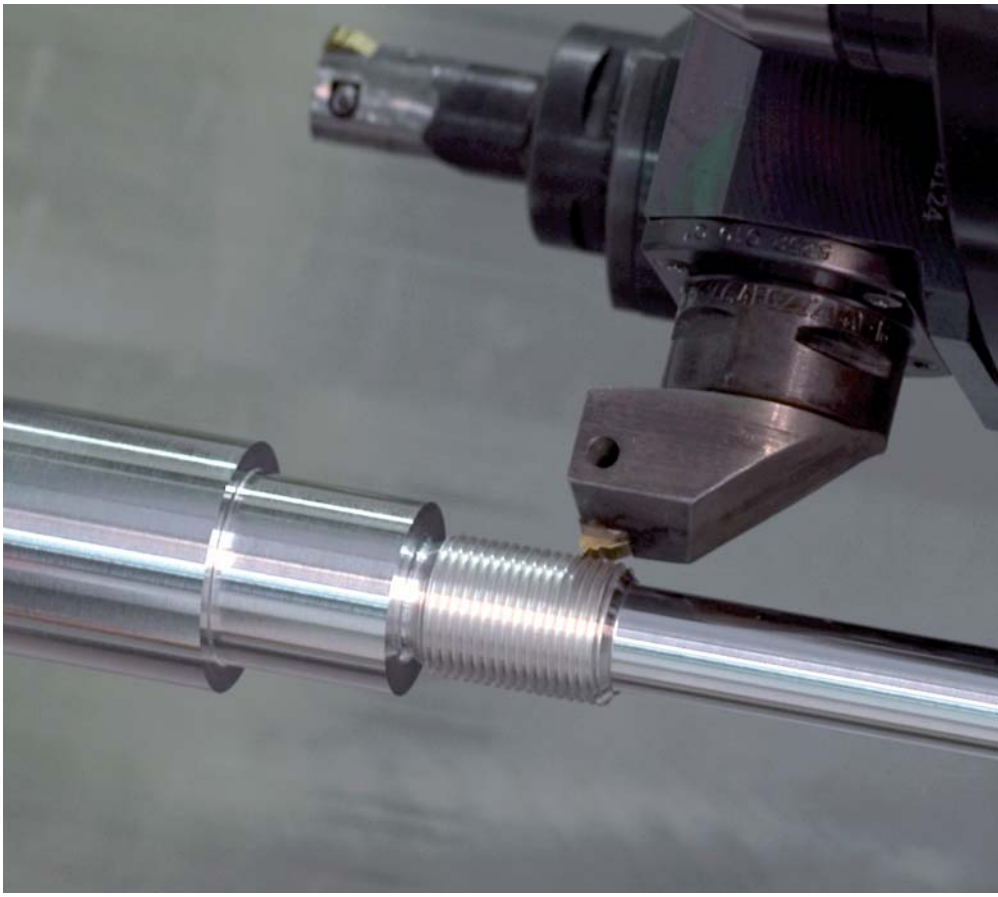
Easy machining with cycles (option)
– Preprogrammed machining steps

On the MANUALplus 620, you can use the handwheel to perform simple operations, such as turning or facing, just like on any conventional lathe. Standard machining operations, such as area clearance, slot milling, recess turning, undercutting, parting, thread cutting, boring, drilling, and milling are stored in the MANUALplus 620 as cycles. You simply enter the positions, dimensions and specifications, and the control will automatically run the machining program.

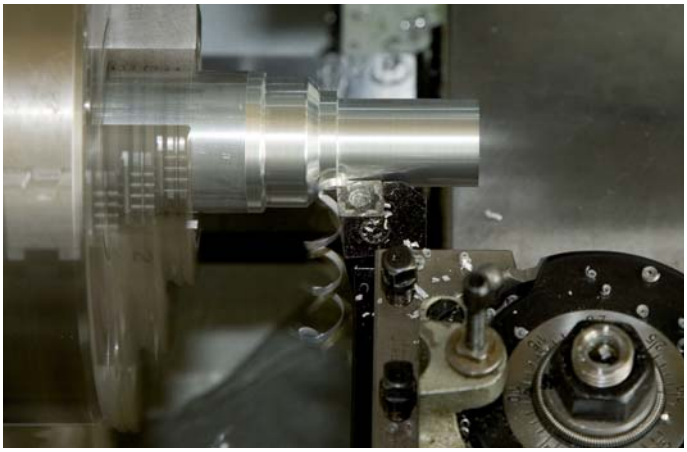
Manual workpiece machining
The MANUALplus 620 simplifies manual turning with numerous functions, without requiring you to learn complicated procedures. This enables you, for example, to adjust the feed rate and spindle speed steplessly during machining, to machine with preset tools or to part with constant cutting speed during machining.

Workpiece machining with cycles
For simple, non-recurring tasks, reworking, thread repair or small production runs, the cycles of the MANUALplus simplify your work.

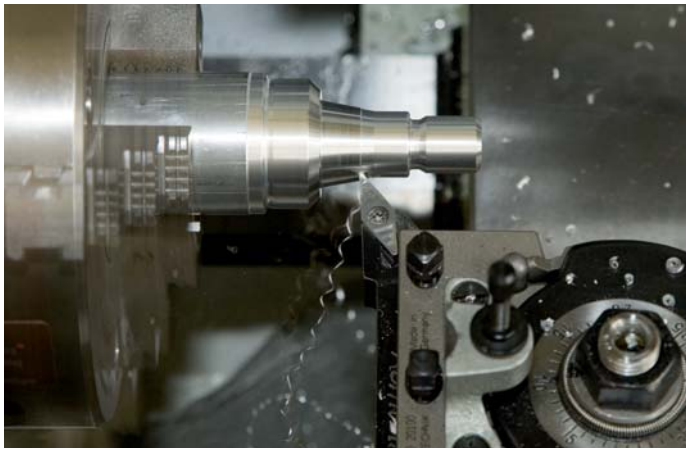
Help graphics and dialogs illustrate the few entries needed for the cycles. Before cutting, use the simulation to assure yourself that the machining will run as planned.



Thread cutting—three times faster with a cycle



Turn cylinders manually:
You work as usual with the handwheels and the position display on the screen.



Turn tapers automatically:
Just enter the dimensions—the MANUALplus 620 moves the saddle and cross-slide automatically.

Fewer calculations
The MANUALplus automatically calculates the number of cuts for roughing, recessing, recess turning or thread cutting, and for pecking it determines the required number of infeeds. When turning a taper, you can enter either the starting point and end point, or the starting point and the taper angle—whichever is shown on your drawing.

Constant availability of tool data
The MANUALplus uses a tool database. Tool data, such as cutting radius, tool angle and point angle only have to be entered once to find the setting dimensions, for example by touch-off. The MANUALplus saves the data. The next time you use the tool, you simply call the tool number. The MANUALplus automatically adjusts for the correct tool size. You can immediately work to dimension.

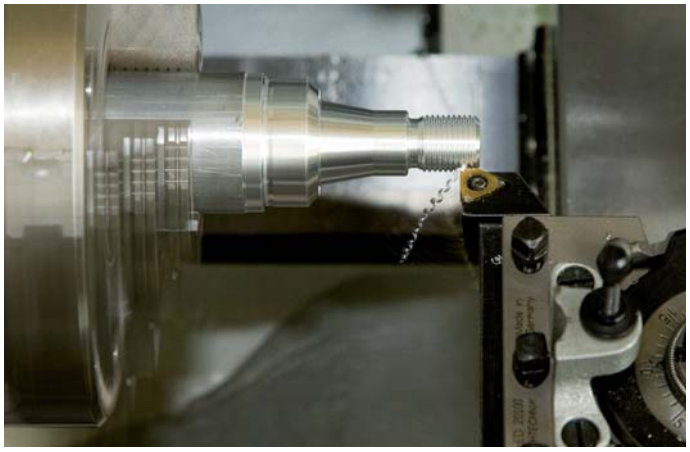
When turning a contour, the MANUALplus automatically compensates the deviations due to the cutting-edge radius. This increases the precision of your workpiece.

Technology data as default values
The MANUALplus saves the cutting data according to the criteria of workpiece material, tool material and machining mode. As you have already entered the cutting material in the tool definition, you need only enter the material of your workpiece. This provides the cycle with all data required for setting default values for the cutting data.

Reference points
You can define the **workpiece datum** by touching the workpiece with the tool or by entering the datum coordinates.

Approach the **tool-change point** once and store this position. Then a simple cycle call suffices to return to the tool change point.

Protective zone for the spindle
For every tool movement in the negative Z direction, the MANUALplus checks whether the programmed protective zone would be violated. If so, it stops the movement and responds with an error message.



Run a cycle to cut a thread automatically:
Call the appropriate fixed cycle and enter the dimensions. The MANUALplus 620 performs the operation automatically.

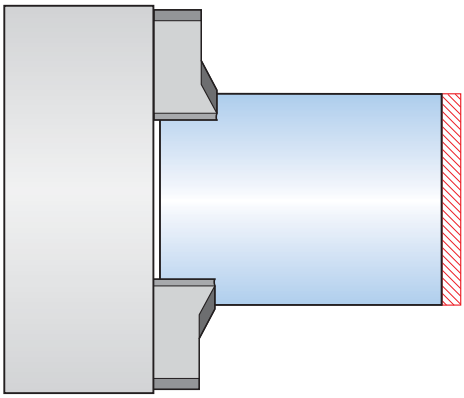
Easy machining with cycles (option)
– From single parts to series

The first workpiece
Machine the workpiece cycle for cycle in the usual way and save the machining steps. When you have finished machining, you save the created cycle program, creating the working plan for the workpiece. The MANUALplus displays the individual machining steps in the proper sequence on the screen.

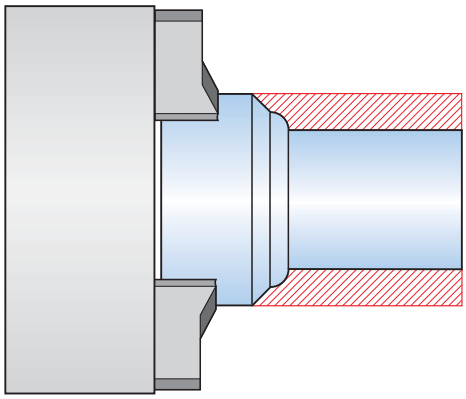
Repeating individual cycles
After you have saved the cycles for one part you can always go back later to edit or delete the steps used to machine it, or insert new steps.

You can save a great deal of time with the MANUALplus when using this feature to produce a family of parts.

Saving your work for the next part
You can store all cycles and rerun them automatically. Each additional part machined saves you time and cost.



1. Transverse turning



2. Longitudinal turning

Cycles for turning
You will always find the appropriate cycle in the cycle menus of the MANUALplus. Help graphics and dialogs explain the operation, all required dimensions and other entries. After entering these values, you can graphically inspect the cutting process and let the operation run automatically.

Threads and undercuts

Single- or multi-start longitudinal, tapered or API thread

Undercuts as per DIN 76, DIN 509 E and DIN 509 F

Undercuts and parting

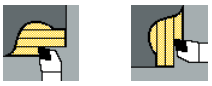
Undercut form H, form K or form U

Parting

Area clearance—cutting and finishing

Longitudinal/transverse cutting for simple contours

Longitudinal/transverse cutting with plunging



Longitudinal/transverse ICP cutting for any contours

Longitudinal/transverse ICP contour parallel cutting

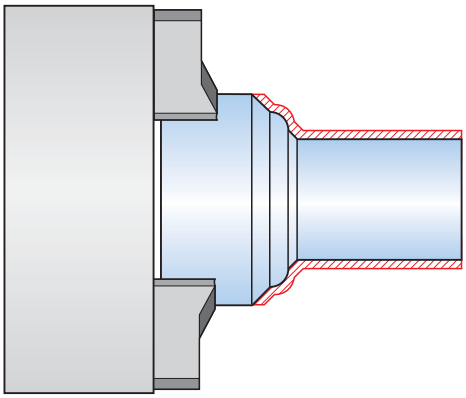
Recessing and recess turning—cutting and finishing

Radial/axial recessing for simple contours

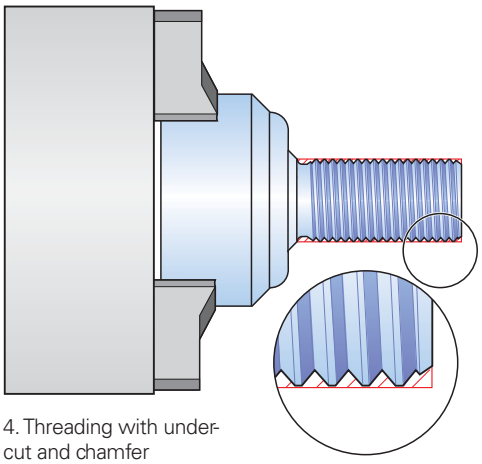
Longitudinal/transverse recess turning for simple contours

Radial/axial ICP recessing for any contours

Longitudinal/transverse ICP recess turning for any contours



3. Finishing



4. Threading with undercut and chamfer

Well thought out, simple and flexible
– Simple programming with smart.Turn (option)

Has the safety clearance been correctly entered, is the speed limit taken into account, how are oversizes defined? All this needs to be considered not only by the beginner, but also by the experienced NC programmer when creating conventional DIN programs.

The smart.Turn principle
The working block—called a unit—plays the central role in smart.Turn programs. A unit describes a machining step completely and unambiguously. The unit includes the tool call, the technology data, the cycle call, the approach and departure strategies as well as global data, such as safety clearance, etc. All these parameters are summarized in one, clearly structured dialog box.

The smart.Turn principle gives you the reassurance that the working block is defined correctly and completely. In the NC program, smart.Turn lists the DIN PLUS commands of the unit. This gives you an overview of all working-block details at any time.

Programming made simple
With smart.Turn, you program with the aid of easy-to-use, unambiguous fillable forms. The overview form shows you a summary of the selected unit, and subforms provide information on the details of a working block. Clearly arranged help graphics illustrate all required input. If input options are available, smart.Turn displays a list of the available options for selection.

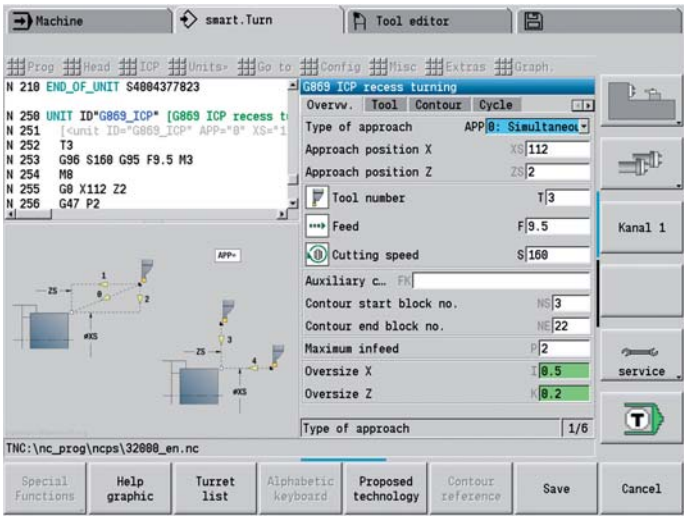
Global program parameters, such as oversizes, safety clearances, coolants, etc., are defined once in the start unit. Then smart.Turn transfers these parameters to the other units.

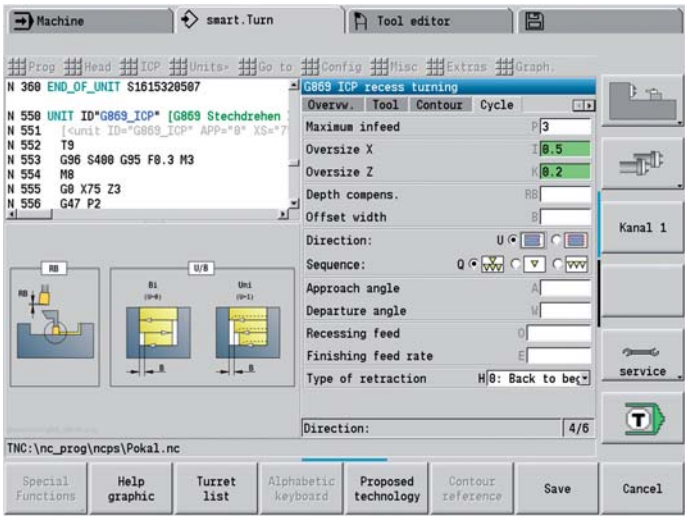
smart.Turn supports units for roughing, finishing, recessing, recess turning, thread cutting, boring, drilling, tapping, and milling, as well as special units for program start, program end, moving the C axis in/out, subprograms and program section repeats.

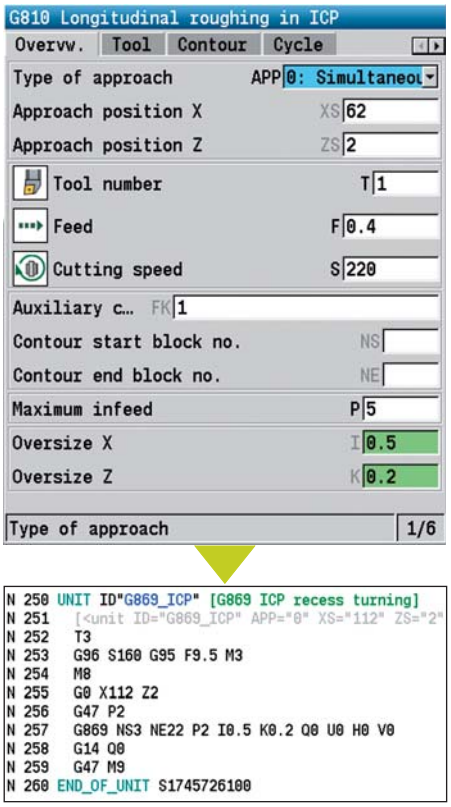
By the way: You do not need to stop the manufacturing process for programming with smart.Turn. You can create and test the smart.Turn program while the program is running.

Structured and easy-to-read
Clearly structured and easy-to-read—these are the characteristics of smart.Turn programs. smart.Turn uses section codes that clearly distinguish between the program head with setup information, the turret assignment, the workpiece description and the actual machining operation.

The smart.Turn technique not only ensures that the program is easy to read, but it also makes it possible to save all information required for producing the workpiece in the NC program.







Programming contours
smart.Turn enables you to work simply and flexibly. Simple contours can be defined with just a few entries in the cycle. Complex contours are described with ICP interactive graphics. Workpiece descriptions that are available in DXF format can be easily imported.

Contour follow-up
When you define the workpiece blank, smart.Turn uses the contour follow-up function, by which the MANUALplus computes the new blank form after each cut. The machining cycles are adapted automatically to the current workpiece blank. The contour follow-up helps you to avoid air cuts and optimize approach paths, even if the workpiece material has been previously removed.

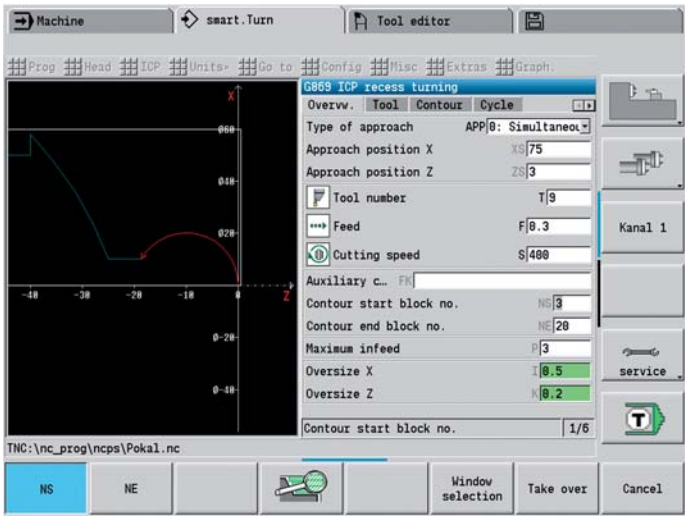
Technology data as default values
The MANUALplus saves the cutting data according to the criteria of workpiece material, tool material and machining mode. As you have already entered the cutting material in the tool definition, you need only enter the material of your workpiece. This provides smart.Turn with all data for setting default values for the cutting data.

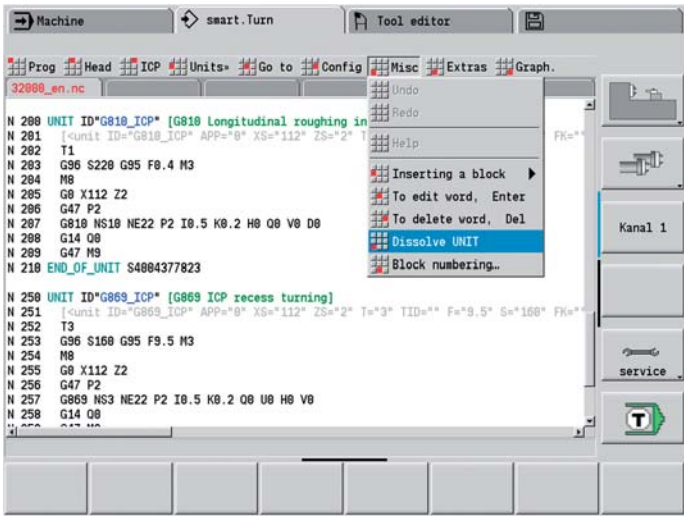
Programming in DIN PLUS
smart.Turn offers units for all machining tasks as well as units for special functions.

If you want to control special machine components, or use the variable programming function or other complex functions that are not provided by smart.Turn, DIN PLUS will support you. It provides powerful machining cycles, program branches and programming with variables. You can switch back and forth between the smart.Turn and DIN PLUS programming modes within a program.

Because the units are based on DIN PLUS, you can break up a unit into blocks at any time to modify and optimize the resulting DIN PLUS program section.

Of course the MANUALplus 620 also allows you to create a DIN program, or to import and use externally created programs.





Well thought out, simple and flexible
– NC program at the push of a button with TURN PLUS (option)

With TURN PLUS you can create part programs in a very short time. After you have entered the contour of the blank and finished part, you only need to select the material and clamping devices. TURN PLUS does everything else automatically: it generates the working plan, selects the machining strategy, selects the tools and cutting data, and generates the NC blocks.

The result is a thoroughly commented smart.Turn program with units that gives you room for optimization and ensures safety when you're breaking in the program.

TURN PLUS can do all that for milling, drilling and boring operations with the C or Y axis on face and cylindrical surfaces as well as on rear-face surfaces in machines with opposing spindles.

The part program at a keystroke
If short programming times are important to you, you can generate all machining steps by pressing a single key. On the basis of the contour entered and the information from the technological database, TURN PLUS independently prepares the working plan and chooses suitable machining strategies, tools and cutting data. The whole operation takes only a few seconds. You can monitor each individual step in the control graphics. TURN PLUS uses a reasonable sequence of possible operations, such as "first roughing transverse, then roughing longitudinal" or "finishing inside, then finishing outside." However, you can adapt this sequence yourself to specific tasks, so the MANUALplus also uses your company's machining know-how for the automatic generation of working plans.

Automatic program generation for full-surface machining
The MANUALplus automatically generates the part program even for complex workpieces that need to be machined on the front face, rear side, and lateral surfaces. After defining the geometry, this can save you about 90 % of the time otherwise needed to create a program.

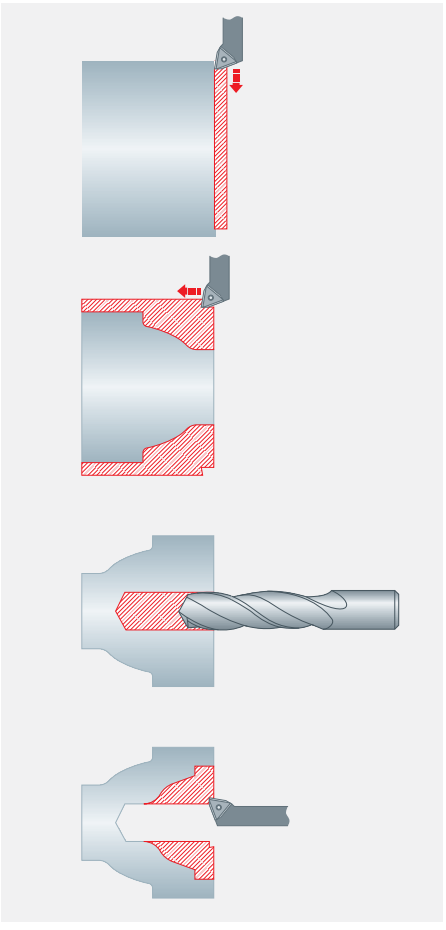
Automatic program generation for the second setup
TURN PLUS knows the contour of the clamping devices when it generates the working plan. It automatically limits the tool path to a safe distance from the clamping device. When the program for the first setup is completed, you can "rechuck" using interactive graphics. The control then automatically generates the program for the second setup using the workpiece geometry that has already been entered.

Go from programming to machining reliably and in the shortest possible time

Inclined contours are no problem
The control is presented with special requirements when it has to generate the part program for inclined contours. Very often the angle of inclination is steeper than the tool tip angle. In that case the control automatically chooses another tool and machines the contour in the opposite direction or as a recess. In any case the result is an executable part program.

Defining the machining sequence
In the TURN PLUS dialog you can define a standard machining sequence.
You can save various machining sequences, e.g for chuck parts or shaft machining.

From the global main types of machining, such as "roughing," "finishing", or "drilling," to details like defining a tool for a specific operation—the automatic working plan generation (AWG) can be adapted to the user's requirements.



Automatic selection of all cycles, tools and cutting data

Describing and importing contours
– ICP interactive contour programming (option)

For jobs that cannot be machined with the standard cycles because of the complexity of the workpiece or the lack of certain dimensions in the workpiece drawing, you need ICP, the interactive contour programming. You describe the contour elements directly as they appear in the workpiece drawing. Or—if the drawing is available in DXF format—you simply import the contour.

Contour programming with ICP
You define an ICP contour by entering the contour elements one after the other in the graphic editor. When selecting the contour elements, you already specify the direction of the line or the direction of rotation of the circular arc. This way the MANUALplus needs very little information about the contour element.

When entering the data, you decide whether the coordinates are absolute or incremental, and whether you enter the end point or the length of the line or the center point or the radius of a circular arc. You also specify whether the path to the next contour element should be tangential or non-tangential.

As long as they are mathematically defined, the MANUALplus calculates missing coordinates, intersections, center points, etc. If the entered data permit several mathematically possible solutions, you can view the individual solutions and then select the proposal that matches the drawing. You can modify or change existing contours.

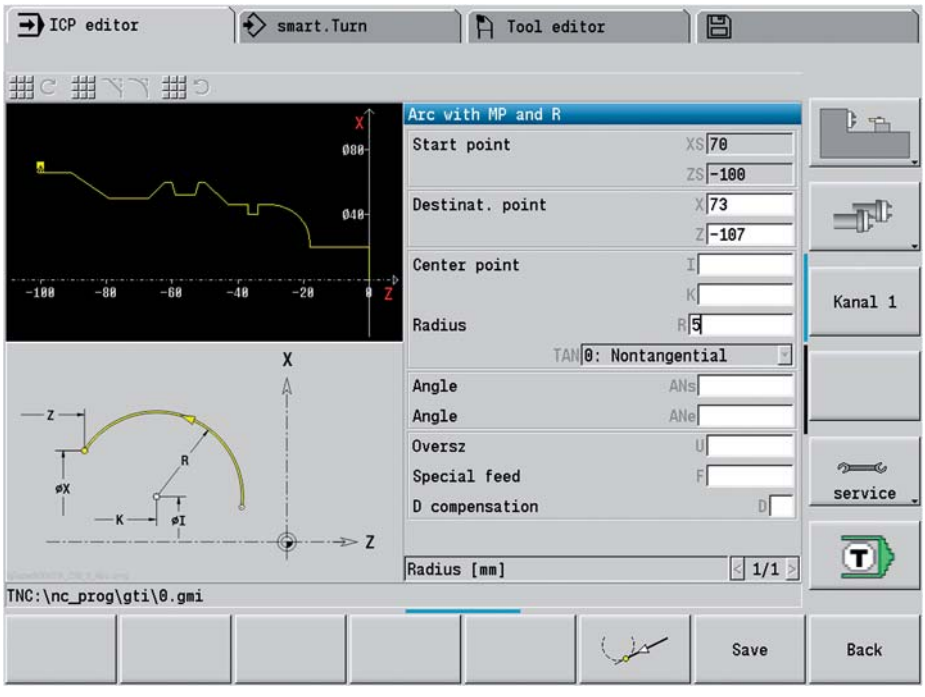
Superimposing form elements
The ICP editor recognizes the chamfer, rounding and undercut (DIN 76, DIN 509 E, DIN 509 F, etc.) form elements. You can enter these form elements in the course of the sequential contour definition. However, it is often easier to first define the "rough" contour, and then to superimpose the form elements. This is done by selecting the corner on which the form element is to be placed and then inserting the element.

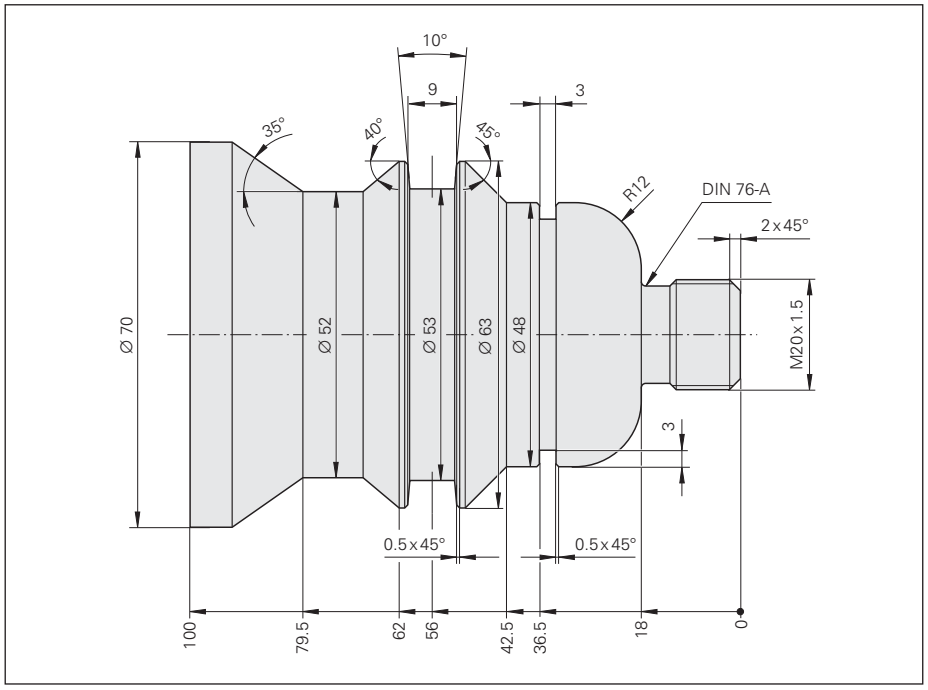
ICP contours for cycle programs
In turning or milling operations, standard contours are defined in the cycle. Complex contours are described with ICP and then called in an ICP roughing, ICP recessing, ICP recess turning or ICP milling cycle.

The ICP editor can be called directly during cycle programming.

ICP contours for smart.Turn and DIN PLUS
In smart.Turn you have various possibilities for describing the contour to be machined. You can describe simple contours right in the unit and use ICP for turning or milling contours as well as linear or circular drilling and milling patterns. The contour defined with ICP is transferred to the smart.Turn program. In the unit you enter a reference to the contour section to be machined.

If you are working in DIN PLUS mode, you can also describe the turning and milling contours, linear and circular patterns with ICP. In the contour-based cycles you enter a reference to the contour section to be machined.





DXF import of contours (option)
Why should you painstakingly enter contour elements if the data already exists in the CAD system? ICP makes it possible to import contours in DXF format directly into the MANUALplus 620. Not only does this save time otherwise spent on programming and testing, but you can also be sure that the finished contour is exactly according to the designer's specifications. DXF contours can describe workpiece blanks, finished parts, contour trains and milling contours. They must exist as two-dimensional elements in a separate layer, i.e. without dimension lines, wrap-around edges, etc.

First, you download the DXF file onto the MANUALplus over the network or use a USB stick. Since the DXF format is fundamentally different from the ICP format, the contour is converted from DXF to ICP format during the import. This contour is then treated as a normal ICP contour, and is available for smart.Turn, DIN PLUS or cycle programming.

Realistic testing before machining
– Graphic simulation

Timely detection of errors is very important for the production or repair of single parts. With its graphic simulation feature, the MANUALplus 620 supports you in checking the program for errors—exactly and with the real dimensions of the contour and cutting edge.

Graphic simulation
Before actual machining, you use the graphic simulation to inspect the
- machining sequence,
- proportioning of cuts,
- and the finished contour.

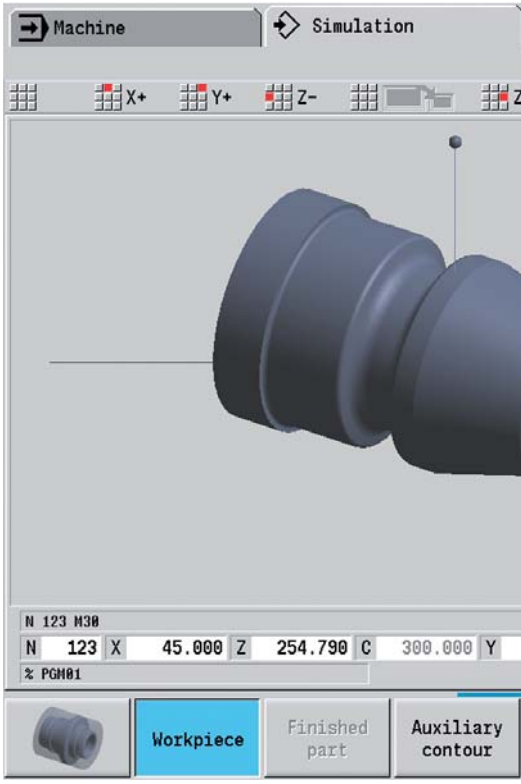
In the graphic simulation you can display the tool cutting edge. You see the cutting-edge radius, the cutting-edge width and the cutting-edge position with their actual dimensions. This helps to recognize machining details or collision risks in time.

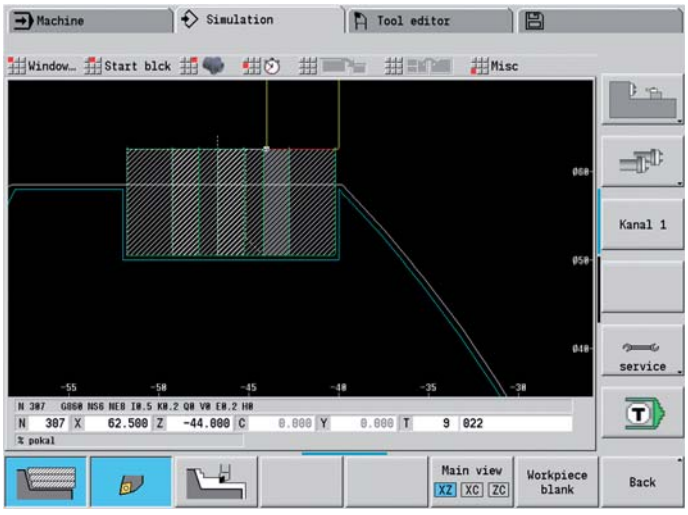
Wire-frame or cutting-path graphics, machining simulation
The MANUALplus supports various views of the tool paths and the machining process. You can choose the type of verification best suited to the tool or machining process used.

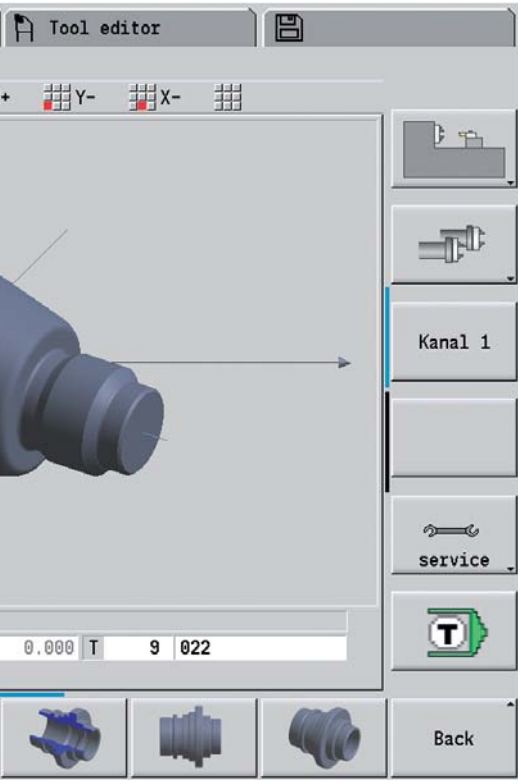
The **wire-frame graphic** is suited for a quick overview of the proportioning of cuts. The path of the theoretical tool tip, however, is not identical with the contour of the workpiece. This view is therefore not as suitable if you wish to run a thorough check of the machined contour.

A more accurate contour verification is provided by the **cutting-path graphics**. The cutting-path graphics account for the exact geometry of the tool tip. You immediately see if material was left behind, the contour is damaged or the overlaps are too large. The cutting-path graphics is especially useful for recessing, drilling and milling operations where the tool shape has an essential influence on the accuracy of the resulting workpiece.

The **machining simulation** (material removal graphic) displays the workpiece blank from which material is removed. The blank is displayed as a white surface. The MANUALplus simulates every tool movement at the programmed cutting speed and removes the material.







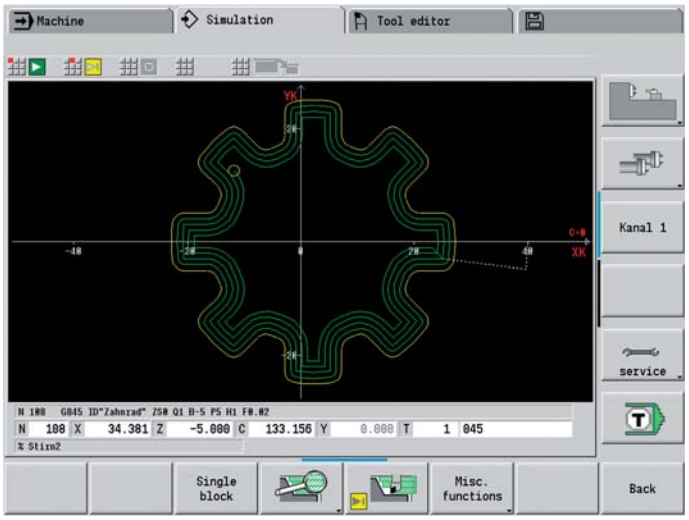
3-D view
With the 3-D graphics the workpiece blank and finished part are shown in a solid-model view. You can rotate the graphic about the principal axes or display it in a sectional view.

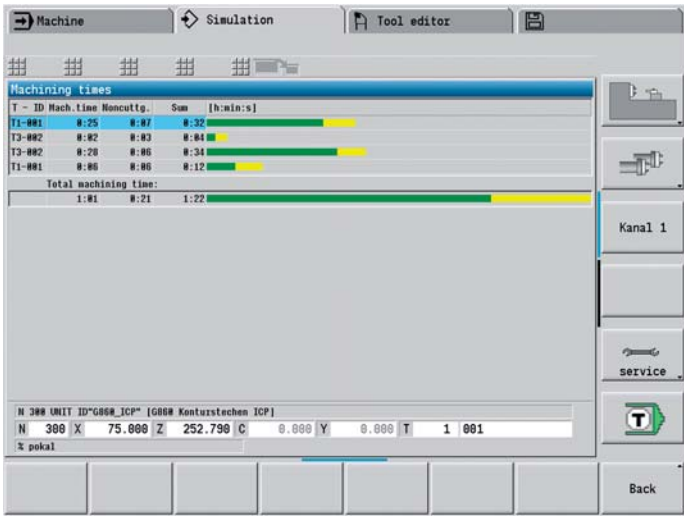
Selecting views
If your lathe is equipped with driven tools and positionable spindle, a C axis or a Y axis, the MANUALplus also simulates machining on the end face and lateral surface, or the XY and YZ plane. You select the combination of windows best suited to the job. This gives you everything you need to closely examine your drilling and milling operations.
The MANUALplus depicts C-axis machining of the cylindrical surface as an "unrolled" plane surface.

Zoom function
Use the "magnifying glass" to set the view to your current needs: greatly magnified in order to control the machining details, or zoomed-out to show the entire workpiece or current working space in order to get an overview of the machining process.

Calculating the machining time
If your customer needs an offer in a hurry, and you need exact information in a very short time, the MANUALplus is a valuable aid with its machining time calculator. During simulation of the cycle, smart.Turn or DIN PLUS program, the MANUALplus calculates the time per piece for the programmed machining.

Along with the total time, the table displays the machining time and idle time of each cycle or each tool insert. This assists you not only in your calculations, but you can also tell at a glance whether there are more possibilities for optimization during the machining process.





Expandable for complex tasks
– Full-surface machining including the C axis and Y axis (option)

The MANUALplus 620 provides a solution for any machining task and any machine configuration: it performs complex machining tasks with a C or Y axis. It also controls full-surface cutting on dual-spindle machines.

And for C-axis, Y-axis, and full-surface machining you can select from the DIN PLUS, smart.Turn or Teach-in programming modes.

C axis or positionable spindle*
For more complex tasks, the MANUALplus 620 can be expanded to also control a C axis or positionable spindle and a driven tool. The driven tool makes it possible to drill off-center and to tap holes while the spindle is at rest. The C axis or positionable spindle permit milling, drilling, and boring on the face and lateral surface of the workpiece. These elements can be displayed for programming and verification in face view and in the unrolled lateral surface view.

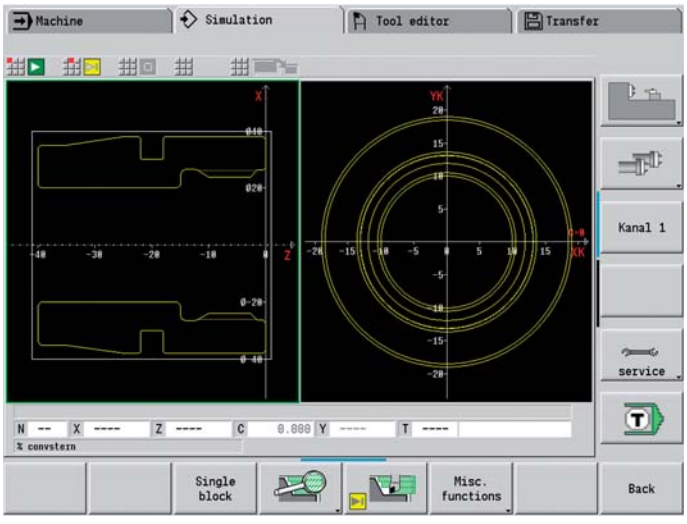
Y axis*
With the Y-axis option of the MANUALplus 620 you can machine slots or pockets with plane bottoms and perpendicular slot walls. By defining the spindle angle, you can determine the position of the milling contours on the workpiece. For programming and verification of these machining sections, the workpiece is shown in side and face view. The Y axis is supported in the smart.Turn and DIN programming feature.

Dual-spindle option
For full-surface lathes, the MANUALplus 620 provides the following features:
- Opposing spindle with second C axis
- Movable tailstock (W axis)

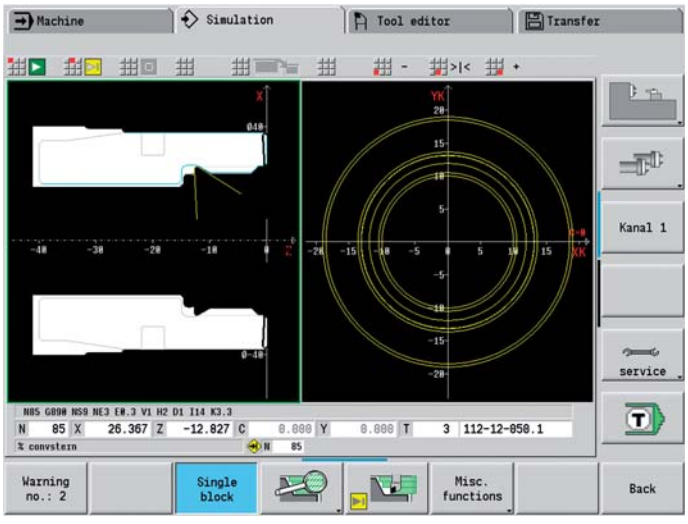
These features are complemented by additional functions such as coordinate transformation, spindle synchronization and traversing to a stop surface.

Coordinate transformation
Contours of workpiece blanks and finished parts can be mirrored about the X axis or shifted relative to the workpiece datum.

* The machine and the MANUALplus 620 must be adapted to this function by the machine tool builder.



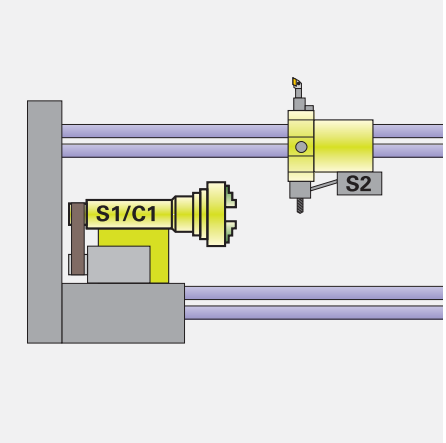
Graphic contour programming for C-axis machining (milling, drilling and boring)



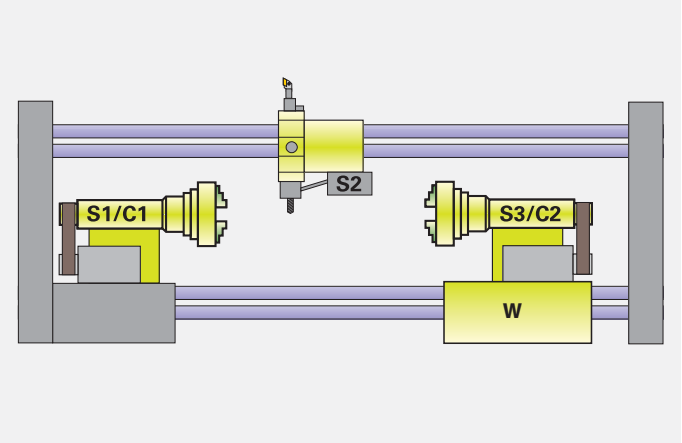
First fixture

Spindle synchronization
Opposing spindles are electronically coupled and rotate synchronously. This makes it possible to transfer the workpiece from one spindle to the other while they are rotating, thereby saving the time otherwise spent braking and starting the spindles. The MANUALplus 620 detects any angular offset and compensates it during subsequent milling on the opposing spindle.

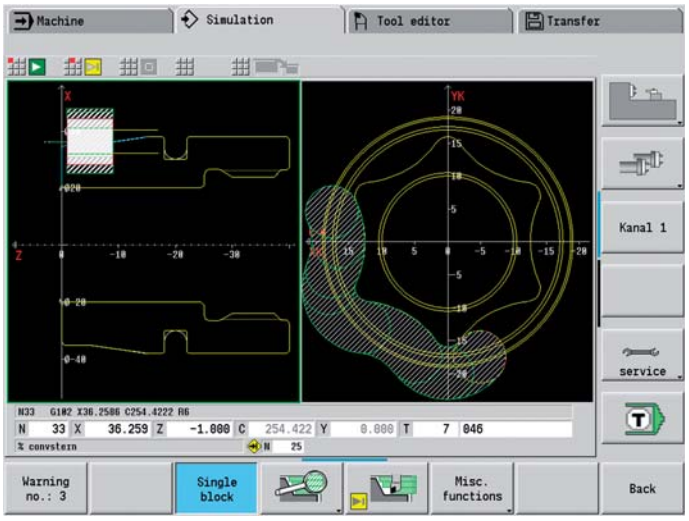
Traversing to a fixed stop
To ensure that the workpiece is firmly pressed to the opposite spindle surface, the control monitors the nominal and actual positions while the longitudinal axis is moving and thereby detects the fixed stop. The MANUALplus monitors the motor torque and uses it to reach the programmed contact force.



Spindle 1 (S1) with C axis (C1) and driven tool (S2)



Full-surface machining: Opposing spindle (S3) with C axis (C2) on secondary axis (W) and driven tool (S2)



Machining of the rear face on the opposing spindle after automatic workpiece transfer

Expandable for complex tasks
– Turning, drilling and milling in one setup (option)

You can use the MANUALplus 620* to drill and mill your workpiece on the end face and lateral surface in one setup with numerous functions and proven cycles.

* Optional. The machine and the MANUALplus 620 must be adapted to this function by the machine tool builder.

Drilling, deep-hole drilling, tapping
The MANUALplus drills, pecks and taps individual holes with the C or Y axis. Via parameters you can easily program infeed reductions for the beginning of drilling or when drilling completely through the workpiece.

Drilling and milling patterns
If bore holes, slots or ICP milling cycles are located at regular distances on a straight line or a circular arc, the MANUALplus greatly simplifies your work: You can create these patterns on the end face or lateral surface with just a few key strokes.

Thread milling
On lathes equipped with a C or Y axis, you can take advantage of thread-milling, because the MANUALplus supports special thread-milling tools.

Milling slots and simple figures
Slot milling with the MANUALplus is very simple. You define the position and depth of the slot as well as the cutting values— the milling cycles automatically take care of the rest.
Even for simple contours such as circles, rectangles and equilateral polygons, just a few keystrokes are necessary to determine the figure and position.

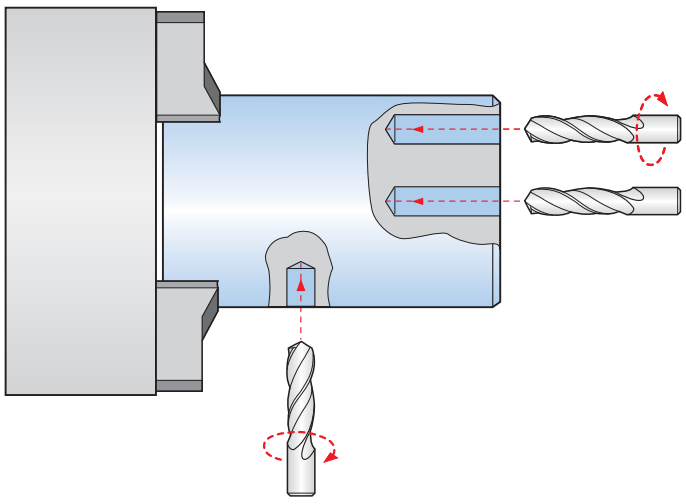
Drilling

Deep-hole drilling

Tapping

Thread milling

Cycles for drilling



Drilling or tapping

Contour and pocket milling
The milling cycles of the MANUALplus support contour and pocket milling. You determine all the important details, such as machining direction, milling direction, approach and departure behavior, infeeds, etc. The MANUALplus automatically compensates for the tool radius.
You can mill the pocket in two stages—first roughing, and then finishing. The result is high accuracy and good surface quality.

In smart.Turn and DIN programming, the MANUALplus 620 supports various infeed strategies. You can choose between direct, reciprocating, or helical infeed, or infeed at the predrilling position.

Face milling
The face milling cycle machines individual surfaces, equilateral polygons or a circle—even off-center.

Helical slot milling
The helical-slot milling cycle is useful for machining lubrication grooves. You specify all important parameters such as pitch, cutting in multiple infeeds, etc.

Engraving cycles
Do you want to "inscribe" your workpieces? That's no problem with the MANUALplus 620. The smart.Turn units for engraving only need a few parameters to engrave characters of any size on a face or lateral surface, or on the XY or YZ plane.

On the workpiece face you can arrange the characters on a line or an arc. On the lateral surface, and when engraving with the Y axis, you define the angle at which the characters are to be arranged

Of course, the engraving cycles are also available as DIN PLUS cycles.

Deburring
The MANUALplus supports special units or DIN PLUS cycles for deburring. You enjoy the benefit of being able to program this operation with only a few parameters.

Slot milling

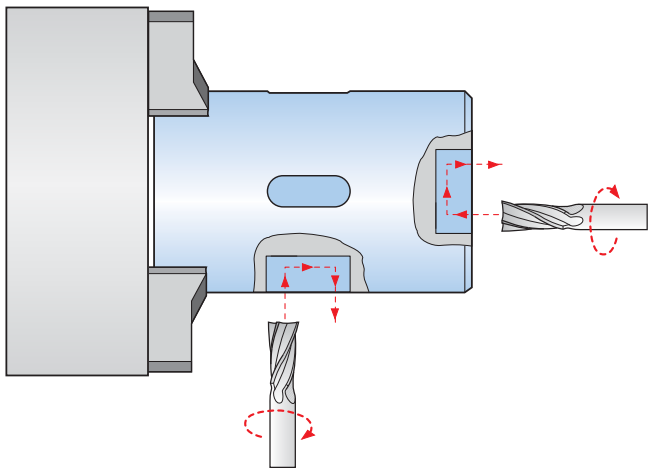
Figure milling (circles, rectangles, regular polygons)

ICP contour milling

Face milling (single surfaces, flattening, polygon)

Helical slot milling

Cycles for milling



Face and lateral-surface milling

Expandable for complex tasks
– Working in a tilted plane with the B axis (option)

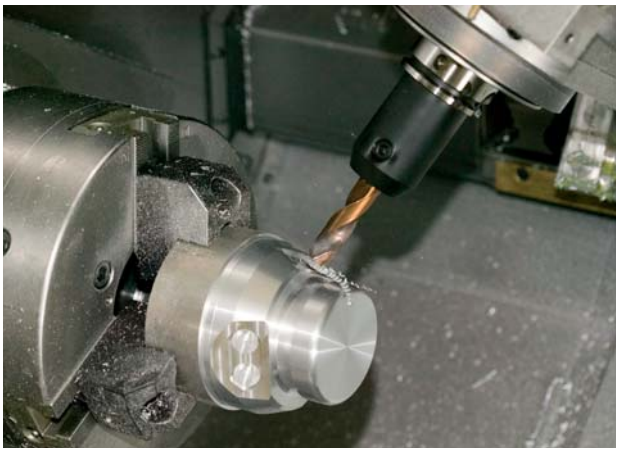
The B axis makes it possible to drill, bore and mill in oblique planes. At first glance, programming for such operations seems very complex and compute-intensive. But with the MANUALplus you simply tilt the coordinate system to the required position and program machining as usual in the working plane. The machine will then execute machining in the tilted working plane.

The B axis also provides benefits for turning operations. By tilting the B axis and rotating the tool you can bring it into positions that enable you to use a single tool to machine in the longitudinal and transverse directions on the main and opposing spindles. That allows you to reduce the number of tools needed as well as do without certain tool changes.

Programming
The usual separation of contour description and machining on the MANUALplus also applies to milling, drilling and boring operations in a tilted plane.

First you rotate and shift the coordinate system so that it lies in the tilted plane. Then you describe the hole pattern or the milling contour as you would in the YZ plane. Here you can use the hole pattern and figure definitions of the MANUALplus. This means that, for linear or circular patterns and simple figures (circles, rectangles, regular polygons, etc.), you only need a few more entries to describe the pattern or figure in the tilted plane.

Simulation
In the Side View window, the simulation shows the hole pattern and milling contour perpendicular to the tilted plane—without distortion. This ensures simple verification of programmed hole patterns and milling contours. You also verify the tool movements in the Side View window. If you want to check the machining in the tilted plane with respect to the rotated contour or the face, add the Lathe Window or Front Window. In the position display (below the simulation window), the MANUALplus displays the angle of the tilted plane and the tilt angle in the B axis. And do you want to see the active coordinate system? No problem—with a simple keystroke the MANUALplus shows the current datum and the direction of the active coordinate system.



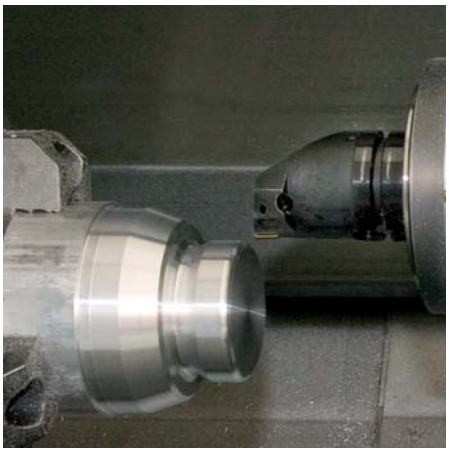
Flexible use of tools*
If your machine is equipped with a B axis, you can use your tools much more efficiently than before. On conventional lathes you need four different tools for longitudinal and transverse turning with opposing spindles. With a B axis, you can do it with a single tool.

You simply tilt the B axis and rotate the tool to the normal position or for machining from behind the workpiece—whichever is required for longitudinal or transverse turning on the main or opposing spindle. All you need is a single call, the MANUALplus calculates the tool lengths, the tool angle and the other tool data for you.

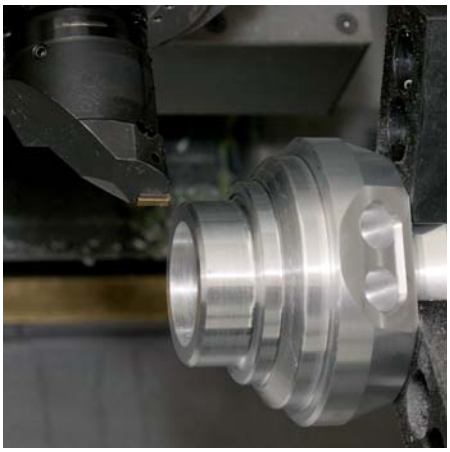
Tool-use flexibility is increased significantly when several tools are mounted in one holder. For example, with a roughing, finishing and recessing tool you can perform considerable parts of turning and recessing operations on a main and opposing spindle—without changing the tool. And programming is very easy. You simply indicate which tooth of the tool to use and then define the tilting angle and the tool position. And no more, because the MANUALplus already has the rest position and the data of each tool tooth in its database.

This type of flexibility lowers the number of your tools, and you save machining time by reducing the number of tool changes.

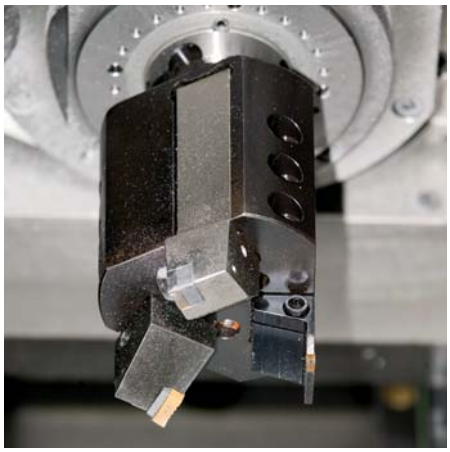
* The machine and MANUALplus must be adapted for this function.



Facing and ...



... longitudinal turning with the same tool ...



... and for several tools on one holder.

Fast availability of tool data and cutting data
– MANUALplus tool database and technology database

Tool database
The MANUALplus can store 250 tools in the standard tool database. The tool database can be expanded to 999 tools (option).
The MANUALplus differentiates between various types of turning, drilling and milling tools. The required data input varies depending on the tool type. In this way you can be sure that all important parameters are specified in spite of reduced data input. The tool data are entered through prompts in which you enter parameters such as cutting-edge radius, tool angle and point angle, cutting material and the tool description. The input parameters are illustrated in context-sensitive help graphics.

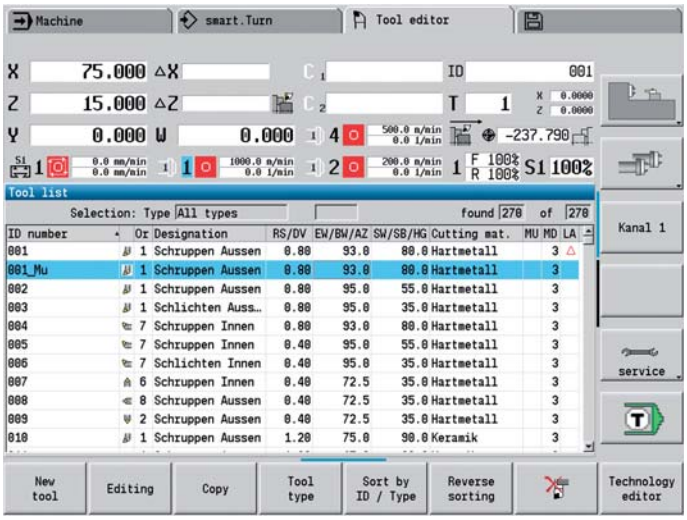
Tool list
The MANUALplus shows all tools in a clearly laid out tool list. Various sorting criteria help you to quickly find the desired tool. This list not only gives you a good overview of your tools—it is also the basis for transferring tool data during manual machining and when you're creating NC programs.

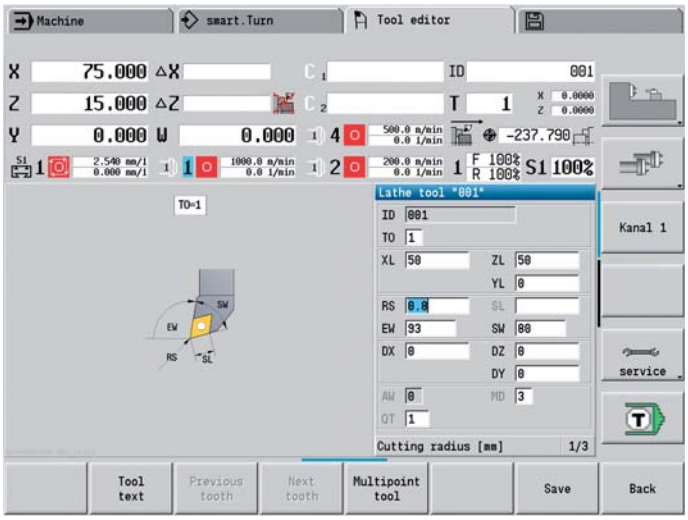
Wear compensation
The MANUALplus offers a simple and straightforward function for compensating tool wear in both the X and the Z axes. You enter the compensation values during or after workpiece machining.

Tool measurement
The MANUALplus 620 offers various possibilities for the measurement of tools directly on the machine:
- By touching the workpiece
- By means of an optical gauge* (option): the tool is manually traversed to the cross hairs of the measuring optics, and the value is saved with a keystroke.
- Through a tool touch probe* (option): the tool moves in measuring direction. The tool setting dimension is ascertained and adopted when the tool touch probe releases a trigger signal, e.g. the TT 140 touch trigger probe with cuboid probe contact.

You can determine the tool data particularly easily, reliably and precisely during tool measurement with an optical gauge or tool touch probe.

* The machine and MANUALplus 620 must be adapted to this function by the machine tool builder.


Machine
smart.Turn
Tool editor
X 75.000 ΔX
Z 15.000 ΔZ
Y 0.000 W 0.000
ID 001
T 1
Tool list
Selection: Type All types
found 270 of 270
ID number Or Designation RS/DV EW/BW/AZ SW/SB/HG Cutting mat. MU MD LA
001 1 Schruppen Aussen 0.80 93.0 80.0 Hartmetall 3
001_Mu 1 Schruppen Aussen 0.80 93.0 80.0 Hartmetall 3
002 1 Schruppen Aussen 0.80 95.0 55.0 Hartmetall 3
003 1 Schlichten Auss… 0.80 95.0 35.0 Hartmetall 3
004 7 Schruppen Innen 0.80 93.0 80.0 Hartmetall 3
005 7 Schruppen Innen 0.40 95.0 55.0 Hartmetall 3
006 7 Schlichten Innen 0.40 95.0 35.0 Hartmetall 3
007 6 Schruppen Innen 0.40 72.5 35.0 Hartmetall 3
008 8 Schruppen Aussen 0.40 72.5 35.0 Hartmetall 3
009 2 Schruppen Aussen 0.40 72.5 35.0 Hartmetall 3
010 1 Schruppen Aussen 1.20 75.0 90.0 Keramik 3
Kanal 1
service
New tool
Editing
Copy
Tool type
Sort by ID / Type
Reverse sorting
Technology editor



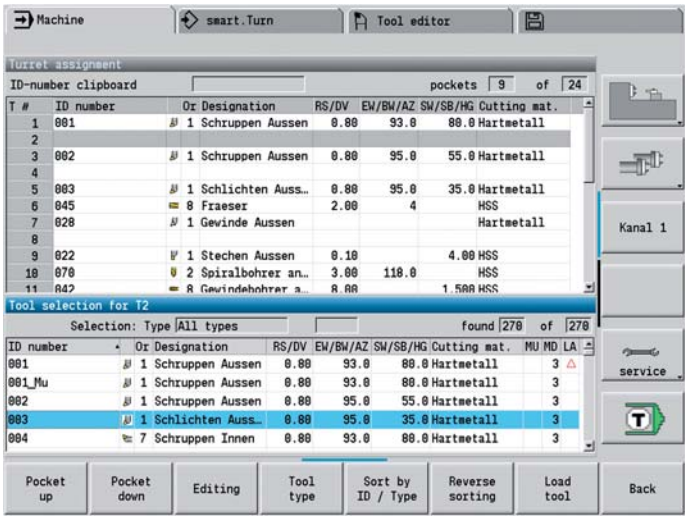
Turret assignment
If your lathe is equipped with a tool turret, you can view the programmed turret assignment at any time. The MANUALplus displays all important tool parameters.

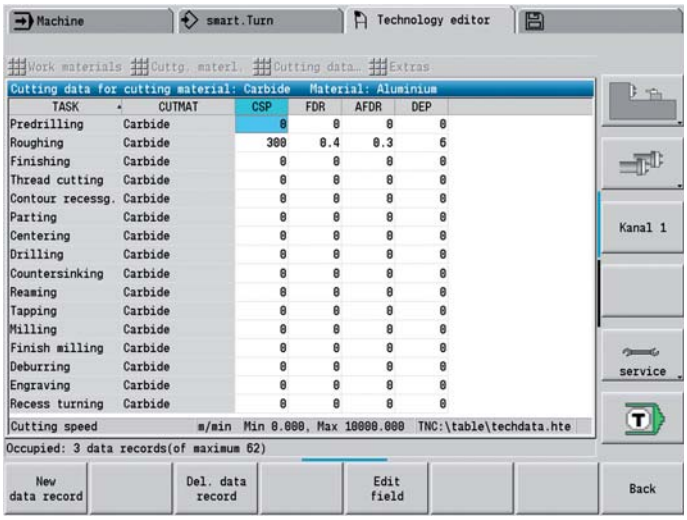
If you want to change the tool assignment or the tools in the turret, you can additionally display the entries of the tool database in the lower window. Now you need only select the desired turret pocket and choose the correct tool from the database. You can transfer the tool data to the turret assignment entry with a simple keystroke.

Technology data (option)
With the MANUALplus 620 you need enter the cutting data only once. The control saves the cutting data according to the criteria of workpiece material, cutting material, and machining mode. Thanks to this three-dimensional table, the control always knows the correct feed rate and the correct cutting speed.

The MANUALplus 620 determines the machining mode from the cycle or the unit. The cutting material is entered during the tool description. You need only define the workpiece material at the beginning of the cycle program or the smart.Turn program, and the MANUALplus will propose the correct values for your machining operation. You can use the suggested cutting parameters or adjust them if required.

In its standard version, you can store the cutting data for 9 workpiece-material/tool-material combinations in the technology database of the MANUALplus. It can be expanded to 62 combinations (option). Each workpiece-material/tool-material combination includes the cutting speed, the main and secondary feed rates, and the infeed for 16 machining modes.


Machine
smart.Turn
Tool editor
Turret assignment
ID-number clipboard
pockets 9 of 24
T # ID number Or Designation RS/DV EW/BW/AZ SW/SB/HG Cutting mat.
1 001 1 Schruppen Aussen 0.80 93.0 80.0 Hartmetall
2
3 002 1 Schruppen Aussen 0.80 95.0 55.0 Hartmetall
4
5 003 1 Schlichten Auss… 0.80 95.0 35.0 Hartmetall
6 045 8 Fraeser 2.00 4 HSS
7 028 1 Gewinde Aussen Hartmetall
8
9 022 1 Stechen Aussen 0.10 4.00 HSS
10 070 2 Spiralbohrer an… 3.00 118.0 HSS
11 042 8 Gewindebohrer a… 8.00 1.500 HSS
Tool selection for T2
Selection: Type All types found 270 of 270
ID number Or Designation RS/DV EW/BW/AZ SW/SB/HG Cutting mat. MU MD LA
001 1 Schruppen Aussen 0.80 93.0 80.0 Hartmetall 3
001_Mu 1 Schruppen Aussen 0.80 93.0 80.0 Hartmetall 3
002 1 Schruppen Aussen 0.80 95.0 55.0 Hartmetall 3
003 1 Schlichten Auss… 0.80 95.0 35.0 Hartmetall 3
004 7 Schruppen Innen 0.80 93.0 80.0 Hartmetall 3
Kanal 1
service
Pocket up
Pocket down
Editing
Tool type
Sort by ID / Type
Reverse sorting
Load tool
Back


Machine
smart.Turn
Technology editor
Work materials
Cuttg. materl.
Cutting data…
Extras
Cutting data for cutting material: Carbide Material: Aluminium
TASK CUTMAT CSP FDR AFDR DEP
Predrilling Carbide 0 0 0 0
Roughing Carbide 300 0.4 0.3 6
Finishing Carbide 0 0 0 0
Thread cutting Carbide 0 0 0 0
Contour recessg. Carbide 0 0 0 0
Parting Carbide 0 0 0 0
Centering Carbide 0 0 0 0
Drilling Carbide 0 0 0 0
Countersinking Carbide 0 0 0 0
Reaming Carbide 0 0 0 0
Tapping Carbide 0 0 0 0
Milling Carbide 0 0 0 0
Finish milling Carbide 0 0 0 0
Deburring Carbide 0 0 0 0
Engraving Carbide 0 0 0 0
Recess turning Carbide 0 0 0 0
Cutting speed m/min Min 0.000, Max 10000.000 TNC:\table\techdata.hte
Occupied: 3 data records(of maximum 62)
Kanal 1
service
New data record
Del. data record
Edit field
Back

Workpiece measurement
– Setup, presetting and measuring with touch trigger probes

Inspecting workpieces for proper machining and dimensional accuracy
The MANUALplus 620 features measuring cycles for checking the geometry of the machined workpieces. For the measuring cycles you simply insert a 3-D touch probe from HEIDENHAIN into the turret in place of a tool:

- Check whether all machining operations were conducted correctly
- Determine infeeds for finishing
- Detect and compensate tool wear
- Check workpiece geometry and sort parts
- Log measured data
- Ascertain machining error trends

Workpiece touch probes from HEIDENHAIN help you to reduce costs in the workshop and in series production: Together with the MANUALplus, touch probes can automatically perform setup, measuring and inspection functions.

The stylus of a TS touch trigger probe is deflected upon contact with a workpiece surface. At that moment the TS generates a trigger signal that, depending on the model, is transmitted either by cable or over an infrared beam to the control.

HEIDENHAIN touch probes* for workpiece measurement are available in different versions. The ruby ball tips are available in several diameters, and the styli in different lengths.

* The touch probes must be interfaced to the MANUALplus by the machine tool builder.

Touch probes with **cable connection for signal transmission** for machines with manual tool change:
TS 220 – TTL version
TS 230 – HTL version

Touch probes with **infrared signal transmission** for machines with automatic tool change:
TS 440 – Compact dimensions
TS 444 – Compact dimensions, battery-free power supply through integrated air turbine generator over central compressed air supply
TS 640 – Standard touch probe with wide-range infrared transmission
TS 740 – High probing accuracy and repeatability, low probing force

TS 440

SE 640

More information about workpiece touch probes is available on the Internet at *www.heidenhain.de* or in the *Touch Probes* brochure or CD.

Tool measurement
– Measuring length, radius and wear directly in the machine

Tool measurement
Exact measurement of the tool dimensions is a decisive factor for ensuring a consistently high level of production quality. HEIDENHAIN offers the TT 140 tool touch probe for this purpose.

Together with the TT 140, the MANUALplus 620 can automatically measure tools while they are in the machine. The MANUALplus 620 saves the results of measurement in a tool table.

The TT 140 tool touch probe is installed in the machine's working space, where it permits tool measurement directly on the machine.

When probing the tool, the contact plate is deflected and a trigger signal is transmitted directly to the MANUALplus. This way you can determine the tool dimensions quickly and easily and, above all, very precisely. The TT 140 tool touch probe is the ideal supplement to improve the efficiency and quality of your production.

TT 140

More information about tool touch probes is available on the Internet at *www.heidenhain.de* or in the *Touch Probes* brochure or CD.

Open for communication
– Fast data transfer with the MANUALplus 620

The networked MANUALplus 620
The MANUALplus 620 can be integrated into networks and connected with PCs, programming stations and other data storage devices. Even in its standard version, the MANUALplus features a latest generation Fast Ethernet interface. The MANUALplus communicates with NFS servers and Windows networks in TCP/IP protocol without needing additional software. The fast data transfer at rates of up to 100 Mbps guarantees very short transfer times.

USB interface
The MANUALplus 620 supports standard memory media with USB interface. Using USB memory media (such as memory sticks), you can quickly and easily exchange DXF contours, ICP contour descriptions, NC programs, tool parameters, etc., between systems that are not connected to each other.

All programs at a glance
After entering the path of the partner terminal, your own MANUALplus programs will be listed on the left side of the screen, and your partner's programs are on the right side. Now select the programs that you want to transfer and press the send or receive button. The data is transferred reliably and almost instantaneously.

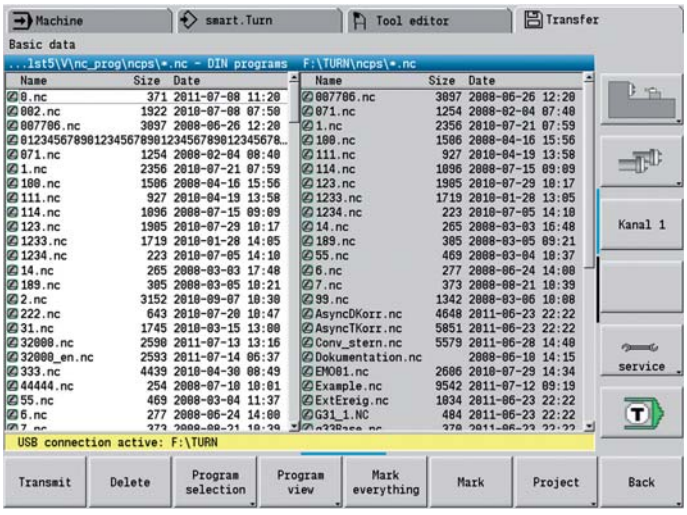
Transferring programs
The easiest and most convenient method of transferring data is to integrate the systems into your company network.

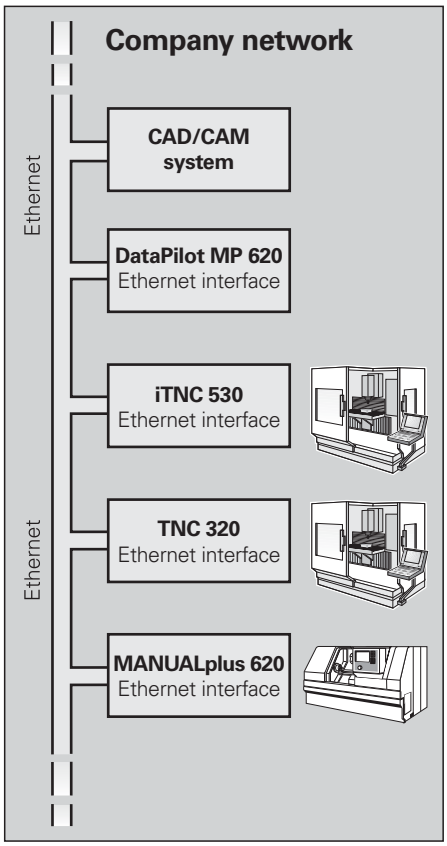
When transferring NC programs, the MANUALplus even considers the files related to the cycle program, smart.Turn program or DIN PLUS program, such as contour descriptions, DIN macros or subprograms.

Exchanging tool data
Once you have acquired tool data, you may also transfer them. That is not only important for data backup: it also helps you when using the PC programming station DataPilot. The benefits: no redundant data acquisition; your files are always up to date.

Programs for data transfer
With the aid of the free PC software **TNCremo** from HEIDENHAIN and an Ethernet or other data interface you can
- transfer remotely stored part programs and tool or pallet tables in both directions,
- make backups.

With the powerful **TNCremoPlus** PC software you can also transfer the screen contents of the control to your PC using the live-screen function.





– Display any file formats on the control screen

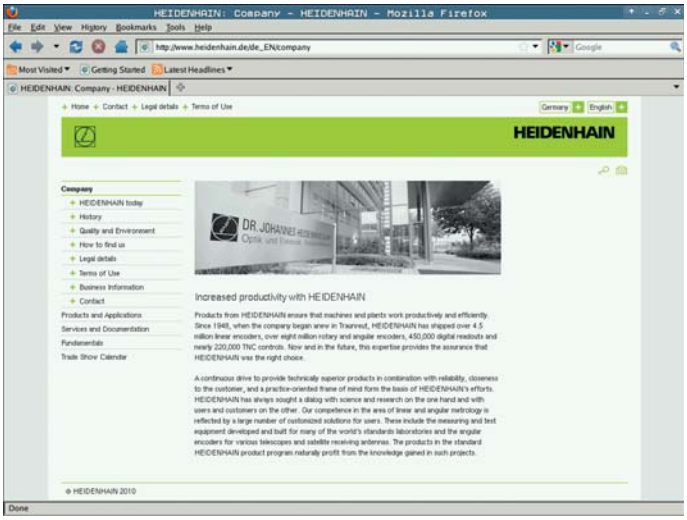
The new integrated PDF viewer enables the user to open PDF files directly on the control. The PDF format is a widely used data format that can be generated out of a great variety of applications. This enables you to easily view work instructions, drawings or other information in the MANUALplus.

The integrated browser now lets you connect the MANUALplus to the Internet and access it directly from the control.

The following further file formats can also be opened directly on the MANUALplus with corresponding editors, and sometimes edited.

- Text files ending with .txt, .ini
- Graphic files ending with .gif, .bmp, .jpg, .png
- Table files ending with .xls and .csv
- HTML files

An external USB pointing device is required for operation.



Open for communication
– The DataPilot MP 620 programming station

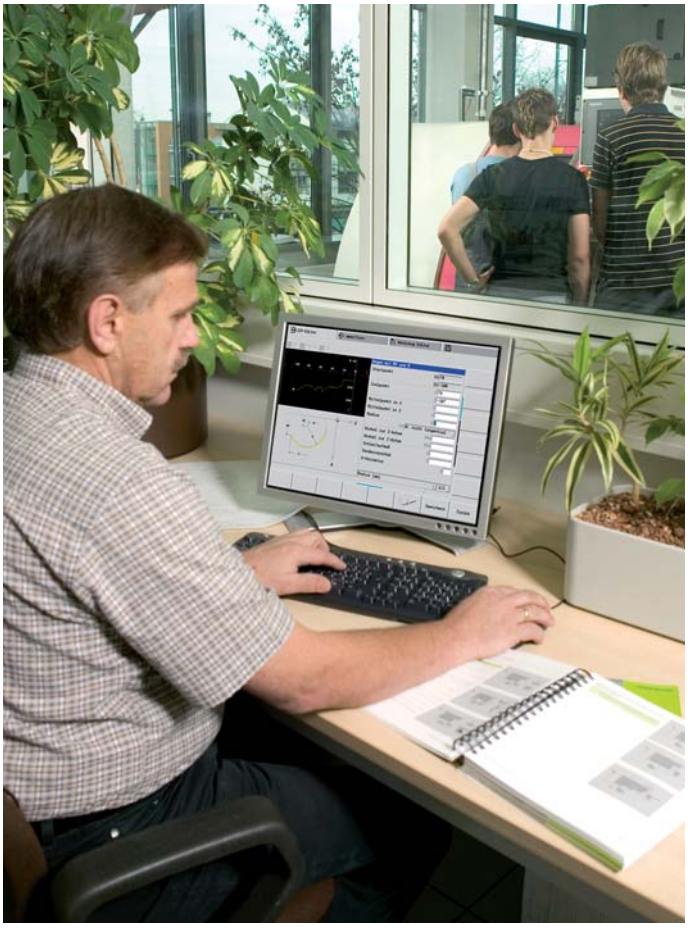
DataPilot MP 620 is the PC programming station for the MANUALplus 620 and the organizing system for the workshop and design office. By shifting the programming, testing and optimization of the program to the PC, machine idle times are greatly reduced.

That is why DataPilot MP 620 is the ideal supplement to the MANUALplus 620 for program creation, archiving, and apprentice and advanced training.

Creating programs
Programming, testing and optimizing cycle programs, smart.Turn programs and DIN PLUS programs with DataPilot on your PC substantially reduce idle machine times. You do not need to adjust your way of thinking, since you program and test with DataPilot in exactly the same way as on the lathe. DataPilot is based on the same software as the control. This ensures that a program created with DataPilot can be run on the machine immediately.

Archiving programs
Even though the MANUALplus has a large memory capacity, you should also back up your programs on an external system. The MANUALplus features a USB and an Ethernet interface. This enables you to integrate the MANUALplus into your existing network or to connect the DataPilot PC directly to the control.

Convenient program transfer functions support both programming as well as archiving on the DataPilot PC.



Training with DataPilot MP 620
Because the DataPilot MP 620 is based on the same software as the MANUALplus 620, it is ideally suited for apprentice and advanced training. Programming and program testing on the DataPilot PC function exactly the same as they do on the machine. DataPilot even simulates setup functions such as defining the workpiece datum, measuring tools or running individual cycles or cycle, smart.Turn or DIN PLUS programs. This gives the trainee the experience needed to enable him or her to safely operate the machine later.

System requirements
DataPilot runs on PCs with the Windows XP, Windows Vista or Windows 7 operating systems.

– Fast availability of all information

Do you have questions on a programming step, but your User's Manual is not at hand? No problem: The MANUALplus 620 and DataPilot PC 620 now feature TURNguide, a convenient help system that can show the user documentation in a separate window.

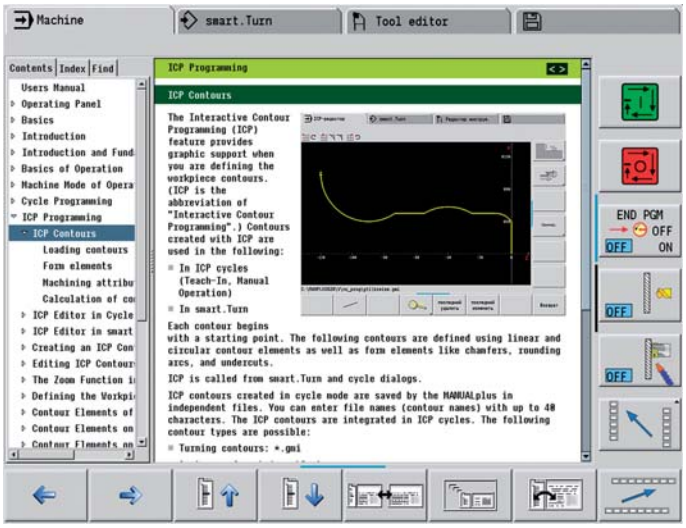
You activate TURNguide simply by pressing the Info key on the keyboard.

TURNguide usually displays the information in the immediate context of the element in question (context-sensitive help). This means that you immediately receive the relevant information. The function is particularly helpful for the programming of cycles. The respective operating method is explained in detail in an open dialog window when you press the Info key.

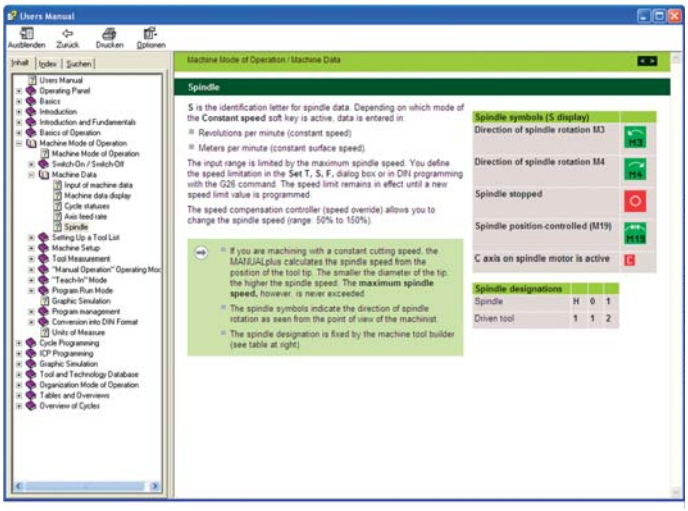
You can download the documentation in the desired language from the HEIDENHAIN homepage into the corresponding language directory on your control.

The following manuals are available in the help system:

- MANUALplus 620 User's Manual
- User's Manual for smart.Turn and DIN programming
- User's Manual for the DataPilot MP 620 (only included in the programming station)



TURNguide integrated in the control, e.g. on the MANUALplus 620 ...



... or at a programming station

Overview
– User functions

User functions	Standard	Option	
Configuration	•		Basic version: X and Z axis, spindle
		0-6	Driven tool and auxiliary axes (U, V, W)
		55+0-6	C axis and driven tool
		70+0-6	Y axis
		54+0-6	B Axis
		94+0-6	Parallel axes (U, V, W)
		132+0-6	Opposing spindle
	•		Digital current and speed control
Operating modes			
Manual operation	•		Manual slide movement through axis-direction keys, intermediate switch or electronic hand-wheels
	•		Graphic support for entering and running cycles without saving the machining steps in alternation with manual machine operation
		11	Thread reworking (thread repair in a second workpiece setup)
Teach-in mode		8	Sequential linking of fixed cycles, where each cycle is run immediately after input, or is graphically simulated and subsequently saved
Program run			All are possible in single-block and full-sequence modes
	•		DIN PLUS programs
		9	smart.Turn programs
		8	Cycle programs
Setup functions	•		Workpiece datum setting
	•		Definition of tool-change position
	•		Definition of protection zone
	•		Definition of machine dimensions
	•		Manual programs
	•		Tool measurement by touching the workpiece
		17	Tool measurement with a TT tool touch probe
		17	Tool measurement with an optical gauge
		17	Workpiece measurement with a TS workpiece touch probe
Programming		8	Turning cycles for simple and complex contours, as well as contours defined with ICP
Cycle programming		8	Contour-parallel turning cycles
		8	Recessing cycles for simple or complex contours, as well as contours defined with ICP
		8	Repetitions with recessing cycles
		8	Recess turning cycles for simple and complex contours, as well as contours defined with ICP
		8	Undercut and parting cycles
		8	Engraving cycles
		8	Threading cycles for single or multi-start longitudinal, taper or API threads, threads with variable pitch
		8+55	Cycles for axial and radial drilling, pecking and tapping operations with the C axis
		8+55	Thread milling with the C axis
		8+55	Axial and radial milling cycles for slots, figures, single surfaces and polygons as well as for complex contours defined with ICP for machining with the C axis
		8+55	Helical slot milling with a (multi-start) C axis
		8+55	Deburring of ICP contours
		8+55	Linear and circular patterns for drilling, boring and milling operations with the C axis
		8	Context-sensitive help graphics
		8	Transfer of cutting values from technology database
		8	Use of DIN macros in cycle programs
		8+9	Conversion of cycle programs to smart.Turn programs

User functions	Standard	Option	
Interactive contour programming (ICP)		8/9	Contour definition with linear and circular contour elements
		8/9	Immediate display of entered contour elements
		8/9	Calculation of missing coordinates, intersections, etc.
		8/9	Graphic display of all solutions for selection by the user if more than one solution is possible
		8/9	Chamfers, rounding arcs and undercuts available as form elements
		8/9	Input of form elements immediately during contour creation or by superimposition later
		8/9	Changes to existing contours can be programmed
		8/9	Machining attributes available for individual contour elements
		8/9+55	C-axis machining on face and lateral surface:
			Description of individual holes and hole patterns (only in smart.Turn)
			Description of figures and figure patterns for milling (only in smart.Turn)
			Creation of freely definable milling contours
		9+70	Y-axis machining on the XY and ZY planes (only in smart.Turn):
			Description of individual holes and hole patterns
			Description of figures and figure patterns for milling
			Creation of freely definable milling contours
		8/9+55+ 70+132	Programming of the rear face for full-surface machining with the C and Y axes
		8/9+42	DXF import: Import of contours for lathe and milling operations
smart.Turn programming		9	Basis is the unit, which is the complete description of a machining block (geometry, technology and cycle data)
		9	Dialog boxes divided into overview and detail forms
		9	Fast navigation between the fillable forms and input groups via the "smart" keys
		9	Context-sensitive help graphics
		9	Start unit with global settings
		9	Transfer of global values from the start unit
		9	Transfer of cutting values from technology database
		9	Units for all turning and recessing operations for simple contours and ICP contours
		9+55/70	Units for boring, drilling and milling operations with the C or Y axis for simple holes, milling contours and drilling and milling patterns as well as those programmed with ICP
		9+55	Activating/deactivating special units for the C axis; subroutines and section repeats
		9+55/70	Verification graphics for blank and finished part and for C and Y axis contours
		9	Turret assignment and other setup information in the smart.Turn program
		9	Parallel programming
		9	Parallel simulation
TURN PLUS		63	Automatic smart.Turn program generation with • Automatic tool selection • Automatic turret assignment • Automatic calculation of cutting data • Automatic generation of machining sequence in all working planes, also for C-axis machining (with option 55) and Y-axis machining (with option 70) • Automatic cutting limitation through chucking equipment • Automatic generation of work blocks for rechucking during full-surface machining • Automatic generation of work blocks for rear-face machining (with option 132)

– User functions

User functions	Standard	Option	
Programming	•		Programming in DIN 66025 format
DIN PLUS programming	•		Extended command format (IF ... THEN ... ELSE ...)
	•		Simple geometry programming (calculation of missing data)
	•		Powerful fixed cycles for turning, recessing, recess turning and thread machining
		55	Powerful fixed cycles for boring, drilling and milling with the C axis
		70	Powerful fixed cycles for boring, drilling and milling with the Y axis
	•		Subroutines
			Technology functions for full-surface machining:
	•		– Moving to a fixed stop
	•		– Parting control
		131/132	– Spindle synchronization
		132	– Converting and mirroring
	•		– Mechatronic tailstock
	•		Programming with variables
		8/9	Contour description with ICP
	•		Program verification graphics for workpiece blank and finished part
	•		Turret assignment and other setup information in the DIN PLUS program
		9	Conversion of smart.Turn units into DIN PLUS command sequences
	•		Parallel programming
	•		Parallel simulation
Program verification graphics	•		Graphic simulation of the cycle process, or of the cycle, smart.Turn or DIN PLUS program
	•		Display of the tool paths as wire-frame or cutting-path graphics, special identification of the rapid-traverse paths
	•		Machining simulation (2-D material-removal graphic)
		55	Side or face view, or 2-D view of cylindrical surface for verification of C-axis machining
	•		Display of programmed contours
		54	View of the tilted plane (B-axis machining)
	•		View of face and YZ plane for verification of Y-axis machining
	•		Three-dimensional display of the workpiece blank and finished part
		132	Simulation of mirrored contours for rear-face machining
	•		Shifting and magnifying functions
	•		Block scan in the simulation
Machining time analysis	•		Calculation of machining time and idle machine time
	•		Consideration of switching commands triggered by the CNC
	•		Representation of time per individual cycle or per tool change

User functions	Standard	Option	
Tool database	•		For 250 tools
		10	For 999 tools
	•		Tool description can be entered for every tool
	•		Automatic inspection of tool-tip position with respect to the contour
	•		Compensation of tool-tip position in the X/Y/Z plane
	•		High-precision adjustment via handwheel, capturing compensation values in the tool table
	•		Automatic tool-tip and cutter radius compensation
	•		Tool monitoring for lifetime of the insert (tool tip) or the number of workpieces produced
		10	Tool monitoring with automatic tool change after expiration tool life
	•		Management of multipoint tools (multiple inserts or reference points)
	•		Support of quick-change systems
	•		Support of chain and plate magazines
Technology database		8/9	Access to cutting data after definition of workpiece material, cutting material and machining mode. The MANUALplus distinguishes between 16 machining modes. Each workpiece-material/tool-material combination includes the cutting speed, the main and secondary feed rates, and the infeed for 16 machining modes.
		8/9	Automatic determination of the machining modes from the cycle or the machining unit
		8/9	Cutting data is entered in the cycle or in the unit as default values
		8/9	9 workpiece-material/tool-material combinations (144 entries)
		10	62 workpiece-material/tool-material combinations (992 entries)
Conversational languages	•		Chinese (simplified), Chinese (traditional), Czech, Danish, Dutch, English, Finnish, French, German, Hungarian, Italian, Polish, Portuguese, Russian, Spanish, Swedish
		41	For more conversational languages, see *Option*

Overview
– Options

Option number	Option	As of NC software 548328	ID	Comment
0 **1** **2** **3** **4** **5** **6**	Additional axis	01 01 01 03 04 04 04	354540-01 353904-01 353905-01 367867-01 367868-01 370291-01 307292-01	**Additional control loops 1 to 7**
8	Software option 1 Teach-in	01	632226-01	**Cycle programming** • Contour description with ICP • Cycle programming • Technology database with 9 workpiece-material/tool-material combinations
9	Software option 2 smart.Turn	01	632227-01	**smart.Turn** • Contour description with ICP • Programming with smart.Turn • Technology database with 9 workpiece-material/tool-material combinations
10	Software option 3 Tools and technology	02	632228-01	**Tools and technology** • Tool database expanded to 999 entries • Technology database expanded to 62 workpiece-material/ tool-material combinations • Tool life monitoring with exchange tools
11	Software option 4 Thread cutting	01	632229-01	**Threads** • Thread recutting • Handwheel superimposition during thread cutting
17	Touch probe functions	02	632230-01	**Tool and workpiece measurement** • Determining tool-setting dimensions with a tool touch probe • Determining tool-setting dimensions with an optical gauge • Automatic workpiece measurement with a workpiece touch probe
24	Gantry axes	04	634621-01	Gantry axes in master-slave torque control
41	Additional language	02	530184-01 530184-02 530184-03 530184-04 530184-06 530184-07 530184-08 530184-09 530184-10	Slovenian Slovak Latvian Norwegian Korean Estonian Turkish Romanian Lithuanian
42	DXF import	02	632231-01	**DXF import** Import of DXF contours
46	Python OEM process	04	579650-01	Python application on the MANUALplus
49	Double-speed axis	01	632223-01	Short control-loop cycle times for direct drives
54	B-axis machining	03	825742-01	**B axis:** Tilting the working plane, rotating the machining position of the tool
55	C-axis machining	01	633944-01	**C-axis machining**
63	TURN PLUS	03	825743-01	**TURN PLUS:** Automatic generation of smart.Turn programs

– Accessories

Option number	Option	As of NC software 548328	ID	Comment
70	Y-axis machining	02	661881-01	**Y-axis machining**
94	Parallel axes	02	679676-01	**Support of parallel axes (U, V, W)** **Combined display of principal axes and secondary axes**
131	Spindle synchronism	04	806270-01	**Synchronization** (of two or more spindles)
132	Opposing spindle	04	806275-01	**Opposing spindle** (spindle synchronism, rear-face machining)
143	Load Adapt. Control	03	800545-01	LAC: Dynamic adaptation of the control parameters

Accessories	
Electronic handwheels	• **HR 180** panel-mounted handwheels with connection to position inputs, plus • One **HR 130** panel-mounted serial handwheel, or one **HR 410** portable serial handwheel
Tool calibration	**TT 140** touch trigger probe with cuboid probe contact
Workpiece measurement	• **TS 220:** Touch trigger probe with cable connection or • **TS 440:** Touch trigger probe with infrared transmission or • **TS 444:** Touch trigger probe with infrared transmission or • **TS 640:** Touch trigger probe with infrared transmission or • **TS 740:** Touch trigger probe with infrared transmission
DataPilot MP 620/CP 620	Control software for PCs for programming, archiving, and training for the MANUALplus 620 • Full version with license for single station or multiple stations • Demo version (free of charge)
Software for PCs	• **TeleService:** Software for remote diagnostics, monitoring, and operation • **CycleDesign:** Software for creating your own cycle structure • **TNCremo:** Software for data transfer—free of charge • **TNCremoPlus:** Software for data transfer with live-screen function

Overview
– Specifications

Specifications	Standard	
Components	•	MC 6110T main computer Integrated operating panel and 12.1-inch TFT color flat-panel display
	•	CC 61xx or UEC 1xx controller units
Operating system	•	HEROS 5 real-time operating system for machine control
Memory	•	2 GB (on CFR compact flash memory card) for NC programs
Input resolution and display step	•	X axis: 0.5 µm, diameter: 1 µm
	•	Z and Y axis: 1 µm
	•	C axis: 0.001°
	•	B axis: 0.001°
	•	U, V, W axis: 1 µm
Interpolation	•	Straight line: in 2 principal axes (max. ±100 m), optional in 3 principal axes
	•	Circle: in 2 axes (radius max. 999 m), optional additional linear interpolation of the third axis
	•	C axis: Interpolation in the linear axes X and Z with the C axis
Feed rate	•	mm/min or mm/rev
	•	Constant surface speed
	•	Max. feed rate (60 000/pole pairs × ball screw pitch) at f_{PWM} = 5 000 Hz
Spindle		Maximum 60 000 rpm (with 2 pole pairs)
Axis feedback control	•	Integrated digital drive control for synchronous and asynchronous motors
	•	Position loop resolution: Signal period of the position encoder/1024
	•	Cycle time of position controller: 0.2 ms
	•	Cycle time of speed controller: 0.2 ms
	•	Cycle time of current controller: minimum 0.05 ms
Error compensation	•	Linear and nonlinear axis error, backlash, reversal peaks during circular movements
	•	Stick-slip friction
Data interfaces	•	100 BaseT Fast Ethernet interface
	•	3 x USB 2.0 (2 x on back, 1 x on front)
Diagnostics	•	Fast and simple troubleshooting through integrated diagnostic aids
Ambient temperature	•	Operation: In electrical cabinet: 5 °C to 40 °C In operating panel: 0 °C to 50 °C
	•	Storage: –20 °C to +60 °C

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 5061
E-mail: info@heidenhain.de

www.heidenhain.de

Vollständige und weitere Adressen siehe www.heidenhain.de
For complete and further addresses see www.heidenhain.de

DE **HEIDENHAIN Vertrieb Deutschland**
83301 Traunreut, Deutschland
☏ 08669 31-3132
FAX 08669 32-3132
E-Mail: hd@heidenhain.de

HEIDENHAIN Technisches Büro Nord
12681 Berlin, Deutschland
☏ 030 54705-240

HEIDENHAIN Technisches Büro Mitte
08468 Heinsdorfergrund, Deutschland
☏ 03765 69544

HEIDENHAIN Technisches Büro West
44379 Dortmund, Deutschland
☏ 0231 618083-0

HEIDENHAIN Technisches Büro Südwest
70771 Leinfelden-Echterdingen, Deutschland
☏ 0711 993395-0

HEIDENHAIN Technisches Büro Südost
83301 Traunreut, Deutschland
☏ 08669 31-1345

AR **NAKASE SRL.**
B1653AOX Villa Ballester, Argentina
www.heidenhain.com.ar

AT **HEIDENHAIN Techn. Büro Österreich**
83301 Traunreut, Germany
www.heidenhain.de

AU **FCR Motion Technology Pty. Ltd**
Laverton North 3026, Australia
E-mail: vicsales@fcrmotion.com

BA Bosnia and Herzegovina → **SL**

BE **HEIDENHAIN NV/SA**
1760 Roosdaal, Belgium
www.heidenhain.be

BG **ESD Bulgaria Ltd.**
Sofia 1172, Bulgaria
www.esd.bg

BR **DIADUR Indústria e Comércio Ltda.**
04763-070 – São Paulo – SP, Brazil
www.heidenhain.com.br

BY **Belarus**
GERTNER Service GmbH
50354 Huerth, Germany
www.gertnergroup.com

CA **HEIDENHAIN CORPORATION**
Mississauga, OntarioL5T2N2, Canada
www.heidenhain.com

CH **HEIDENHAIN (SCHWEIZ) AG**
8603 Schwerzenbach, Switzerland
www.heidenhain.ch

CN **DR. JOHANNES HEIDENHAIN (CHINA) Co., Ltd.**
Beijing 101312, China
www.heidenhain.com.cn

CZ **HEIDENHAIN s.r.o.**
102 00 Praha 10, Czech Republic
www.heidenhain.cz

DK **TP TEKNIK A/S**
2670 Greve, Denmark
www.tp-gruppen.dk

ES **FARRESA ELECTRONICA S.A.**
08028 Barcelona, Spain
www.farresa.es

FI **HEIDENHAIN Scandinavia AB**
02770 Espoo, Finland
www.heidenhain.fi

FR **HEIDENHAIN FRANCE sarl**
92310 Sèvres, France
www.heidenhain.fr

GB **HEIDENHAIN (G.B.) Limited**
Burgess Hill RH15 9RD, United Kingdom
www.heidenhain.co.uk

GR **MB Milionis Vassilis**
17341 Athens, Greece
www.heidenhain.gr

HK **HEIDENHAIN LTD**
Kowloon, Hong Kong
E-mail: sales@heidenhain.com.hk

HR Croatia → **SL**

HU **HEIDENHAIN Kereskedelmi Képviselet**
1239 Budapest, Hungary
www.heidenhain.hu

ID **PT Servitama Era Toolsindo**
Jakarta 13930, Indonesia
E-mail: ptset@group.gts.co.id

IL **NEUMO VARGUS MARKETING LTD.**
Tel Aviv 61570, Israel
E-mail: neumo@neumo-vargus.co.il

IN **HEIDENHAIN Optics & Electronics India Private Limited**
Chetpet, Chennai 600 031, India
www.heidenhain.in

IT **HEIDENHAIN ITALIANA S.r.l.**
20128 Milano, Italy
www.heidenhain.it

JP **HEIDENHAIN K.K.**
Tokyo 102-0083, Japan
www.heidenhain.co.jp

KR **HEIDENHAIN Korea LTD.**
Gasan-Dong, Seoul, Korea 153-782
www.heidenhain.co.kr

ME Montenegro → **SL**

MK Macedonia → **BG**

MX **HEIDENHAIN CORPORATION MEXICO**
20235 Aguascalientes, Ags., Mexico
E-mail: info@heidenhain.com

MY **ISOSERVE Sdn. Bhd**
56100 Kuala Lumpur, Malaysia
E-mail: isoserve@po.jaring.my

NL **HEIDENHAIN NEDERLAND B.V.**
6716 BM Ede, Netherlands
www.heidenhain.nl

NO **HEIDENHAIN Scandinavia AB**
7300 Orkanger, Norway
www.heidenhain.no

PH **Machinebanks` Corporation**
Quezon City, Philippines 1113
E-mail: info@machinebanks.com

PL **APS**
02-384 Warszawa, Poland
www.heidenhain.pl

PT **FARRESA ELECTRÓNICA, LDA.**
4470 - 177 Maia, Portugal
www.farresa.pt

RO **HEIDENHAIN Reprezentanţă Romania**
Braşov, 500407, Romania
www.heidenhain.ro

RS Serbia → **BG**

RU **OOO HEIDENHAIN**
125315 Moscow, Russia
www.heidenhain.ru

SE **HEIDENHAIN Scandinavia AB**
12739 Skärholmen, Sweden
www.heidenhain.se

SG **HEIDENHAIN PACIFIC PTE LTD.**
Singapore 408593
www.heidenhain.com.sg

SK **KOPRETINA TN s.r.o.**
91101 Trencin, Slovakia
www.kopretina.sk

SL **Posredništvo HEIDENHAIN NAVO d.o.o.**
2000 Maribor, Slovenia
www.heidenhain-hubl.si

TH **HEIDENHAIN (THAILAND) LTD**
Bangkok 10250, Thailand
www.heidenhain.co.th

TR **T&M Mühendislik San. ve Tic. LTD. ŞTİ.**
34728 Ümraniye-Istanbul, Turkey
www.heidenhain.com.tr

TW **HEIDENHAIN Co., Ltd.**
Taichung 40768, Taiwan R.O.C.
www.heidenhain.com.tw

UA **Gertner Service GmbH Büro Kiev**
01133 Kiev, Ukraine
www.gertnergroup.com

US **HEIDENHAIN CORPORATION**
Schaumburg, IL 60173-5337, USA
www.heidenhain.com

VE **Maquinaria Diekmann S.A.**
Caracas, 1040-A, Venezuela
E-mail: purchase@diekmann.com.ve

VN **AMS Co. Ltd**
HCM City, Vietnam
E-mail: davidgoh@amsvn.com

ZA **MAFEMA SALES SERVICES C.C.**
Midrand 1685, South Africa
www.heidenhain.co.za

Zum Abheften hier falzen! / *Fold here for filing!*

634865-24 · 10 · 9/2012 · F&W · Printed in Germany